Student Notes:

CATIA - Virtual Design Training
Foils

Numerical Control Infrastructure

Version 5 Release 20
September 2010

EDU_CAT_EN_NCI_FF_V5R20

Course Presentation

Course objectives

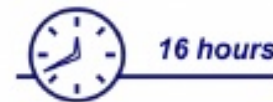**Upon completion of this course you will be able to:**
- **Identify and use the Manufacturing workbenches' tools**
- **Create a Manufacturing Program**
- **Simulate a Manufacturing Program**
- **Manage Tools and Tool Catalogs**
- **Define and verify the Tool Path**
- **Generate NC data using an integrated Post Processor**
- **Create shop floor documentation**
- **Manage design changes**
- **Import V4 data**

Target audience

NC Programmers

Prerequisites

Students attending this course should be familiar with CATIA V5 fundamentals

16 hours

Table of Contents (1/3)

Table of Contents (2/3)

Table of Contents (3/3)

Student Notes:

How to Use This Course

To assist in the presentation and learning process, the course has been structured as follows:

Lessons:
Lessons provide the key concepts, methodologies, and basic skill practice exercises. The goal of each lesson is to present the necessary knowledge and skills to master a basic level of understanding for a given topic.

A Master Exercise:
A Master Exercise provides a project where an industry type part is used to assist you in applying the key knowledge and skills acquired in the individual lessons as they apply to real world scenarios. The master exercise also highlights the process and steps for completing industry parts.

Advanced Topics and Advanced Exercises:
Advanced Topics are covered after above Common Topics and respective exercises are followed after them.

Note: The individual steps of the Master Project may be completed either at the end of the course or at the end of each lesson.
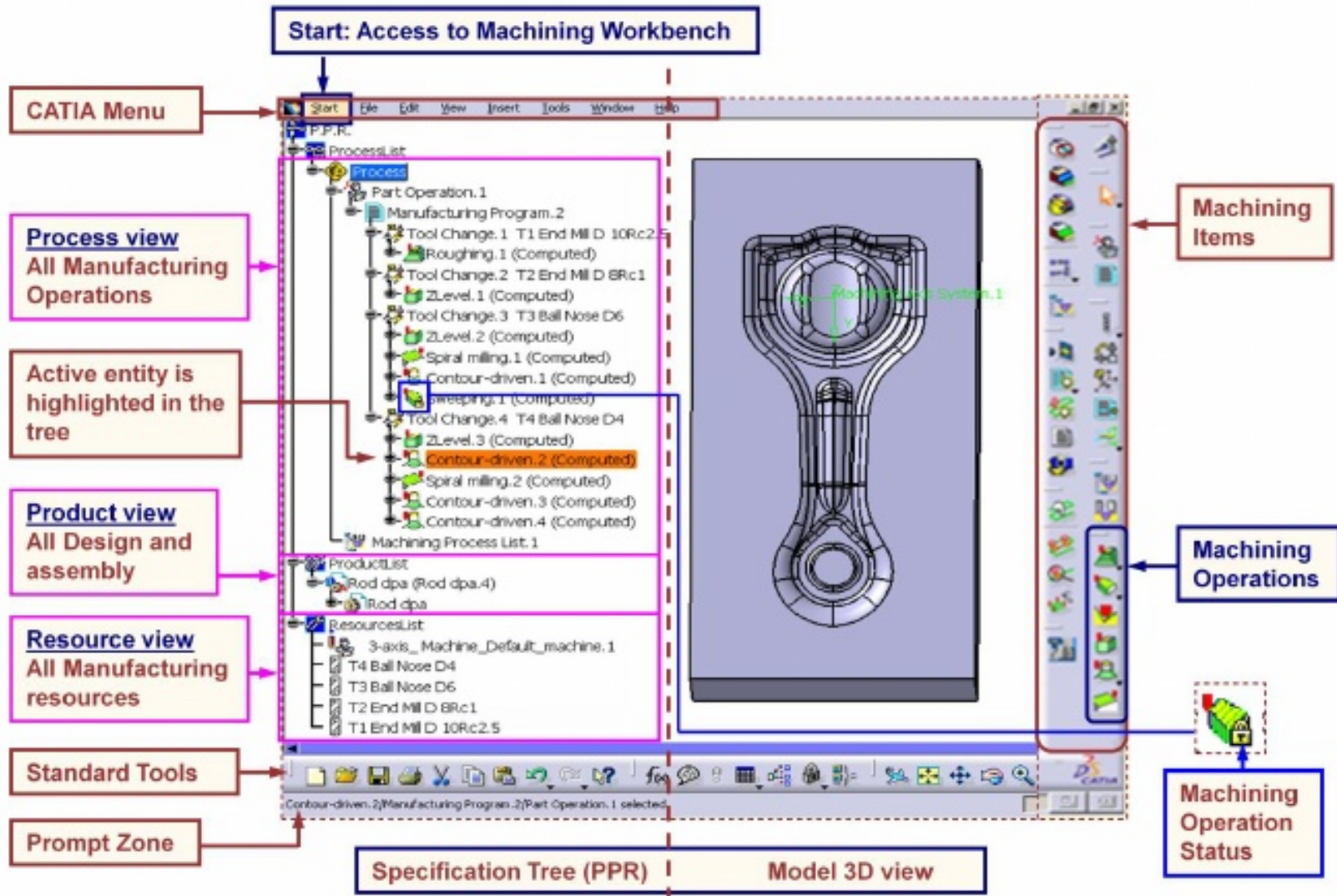
Student Notes:

Manufacturing Workbench Presentation

This lesson covers the following topics:

- Workbench Introduction
- Process Presentation
- Manufacturing Terminology
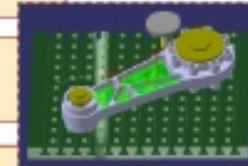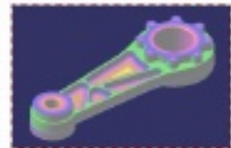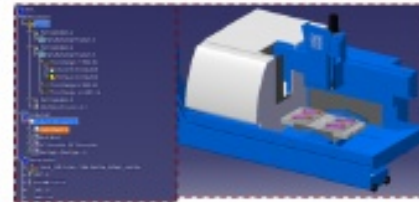- Manufacturing Workbench More Details

Workbench Introduction

Start: Access to Machining Workbench

CATIA Menu

Process view
All Manufacturing Operations

Active entity is highlighted in the tree

Product view
All Design and assembly

Resource view
All Manufacturing resources

Standard Tools

Prompt Zone

Machining Items

Machining Operations

Machining Operation Status

Specification Tree (PPR)   **Model 3D view**

P.P.R.
ProcessList
Process
Part Operation.1
Manufacturing Program.2
Tool Change.1 T1 End Mill D 10Rc2.5
Roughing.1 (Computed)
Tool Change.2 T2 End Mill D 8Rc1
ZLevel.1 (Computed)
Tool Change.3 T3 Ball Nose D6
ZLevel.2 (Computed)
Spiral milling.1 (Computed)
Contour-driven.1 (Computed)
Sweeping.1 (Computed)
Tool Change.4 T4 Ball Nose D4
ZLevel.3 (Computed)
Contour-driven.2 (Computed)
Spiral milling.2 (Computed)
Contour-driven.3 (Computed)
Contour-driven.4 (Computed)
Machining Process List.1
ProductList
Rod dpa (Rod dpa.4)
Rod dpa
ResourcesList
3-axis_ Machine_Default_machine.1
T4 Ball Nose D4
T3 Ball Nose D6
T2 End Mill D 8Rc1
T1 End Mill D 10Rc2.5

Contour-driven.2/Manufacturing Program.2/Part Operation.1 selected

8

Process Presentation

Define NC Assembly: ①
- Part
- Stock
- Fixtures
- Added Geometry (Profiles, Safety Planes, Axis, Points, etc)

Define Setup: ②

Lathe Machining Surface Machining Prismatic Machining

- Access to Machining Workbench
- Define Part Operations necessary to machine all the part

Add Geometry
- in the specified CATPart (if necessary)
- Planes, points

Create Machining Program:
- Create Machining Operations
- Simulate them

③

Simulate & analyze
- Tool Path
- Removal material
- accessibility
- Tool clashes

Re-order the operations
- auto– sequencing (If needed)

Generate Auxiliary Operations
- Rotables,
- PP instructions

NC Data generation ④
- APT, NC Code
- Shop floor Documentation (Text)

Manufacturing Terminology

Part Operation:

A Part Operation (or PO) 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.

Manufacturing Program:

A Manufacturing Program describes the processing order of the NC entities that are taken into account for tool path computation: Machining Operations, Auxiliary Operations.
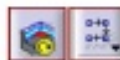
Machining Operation:

A Machining Operation (or MO) contains all the necessary information for machining a part of a work

piece using a single tool. (Such as Drilling, Pocketing, Roughing, Sweeping)

Machining Process:

Group of Machining operation. You can store it in catalog and import it in your current session.

Machining Features:

It's predefined set of geometry that you can directly select in the Machining operation or assign when you instantiate a Machining Process.

Auxiliary Operation:

A control function such as Tool Change or Machine Table/Head Rotation. These commands may be interpreted by a specific Post-processor.

Student Notes :

Manufacturing Workbench More Details

You will learn more details of Manufacturing Workbench.

- Accessing Workbench
- The Process Product Resources Model
- Files Management
- CATProcess Management
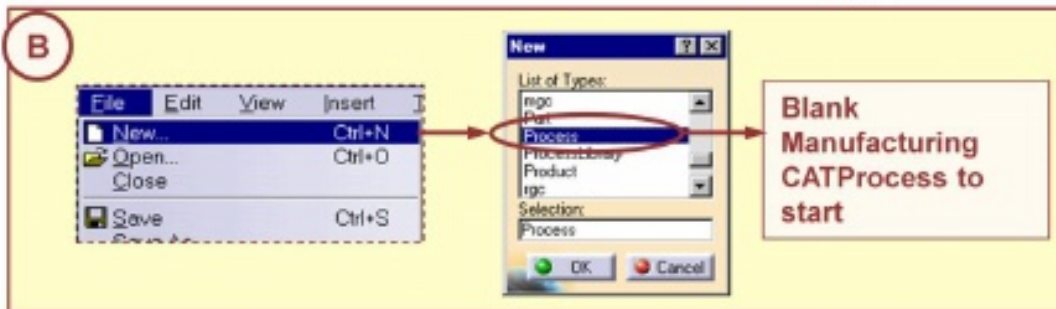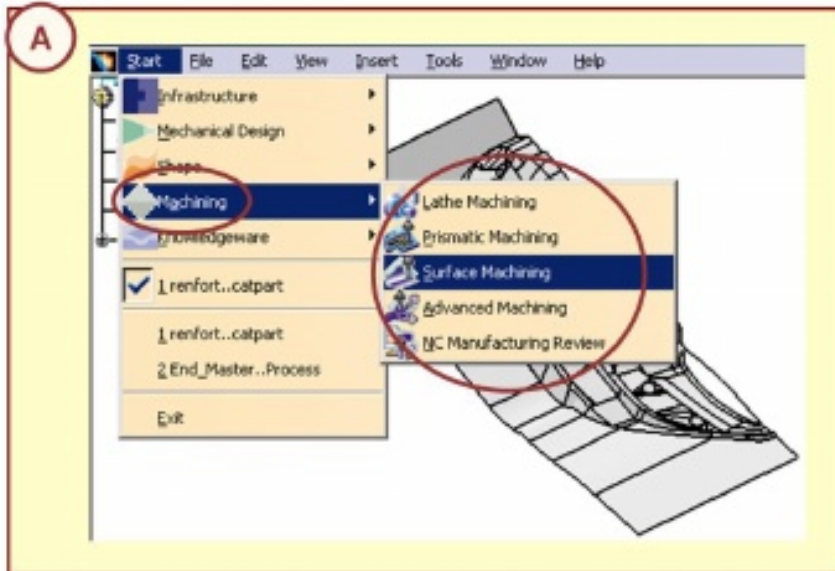- Status of the Machining Operations

Accessing Workbench

Three different ways to access the workbench are:

Anywhere from:
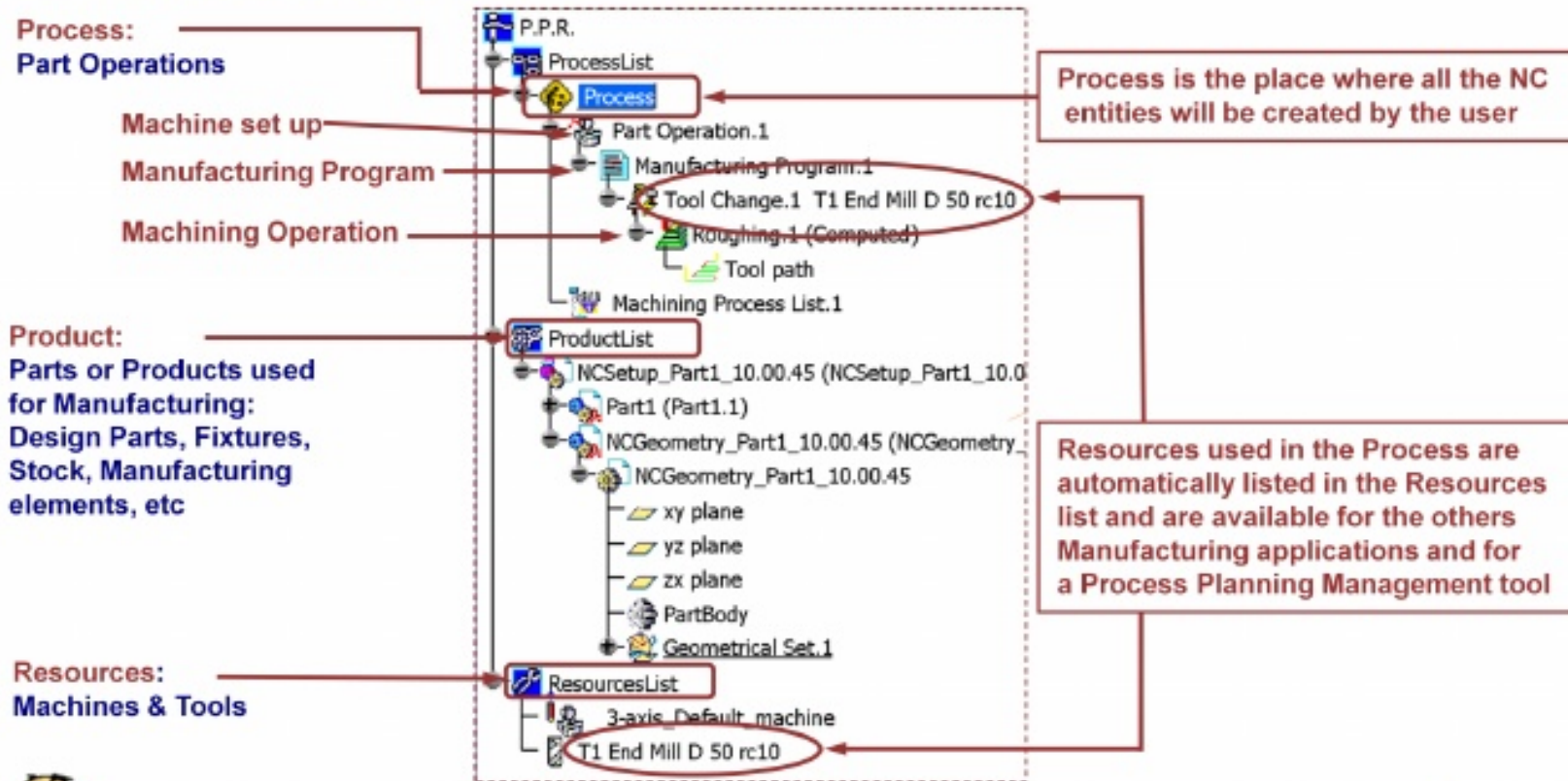A - Start menu or
B - File menu + New or
C - Workbench Icon

(A)

Start File Edit View Insert Tools Window Help

Infrastructure
Mechanical Design
Shape
Machining
Knowledgeware

Lathe Machining
Prismatic Machining
Surface Machining
Advanced Machining
NC Manufacturing Review

1 renfort..catpart
1 renfort..catpart
2 End_Master..Process

Exit

(C)

Welcome to CATIA V5

Lathe Machining Surface Machining Prismatic Machining

Do not show this dialog at startup   Close

See Tools +
Customize + Start
menu for the
content of this
Welcome Box

(B)

File Edit View Insert T
New...   Ctrl+N
Open...   Ctrl+O
Close
Save   Ctrl+S

New
List of Types:
mgc
Part
Process
ProcessLibrary
Product
rgc
Selection:
Process
OK   Cancel

Blank
Manufacturing
CATProcess to
start

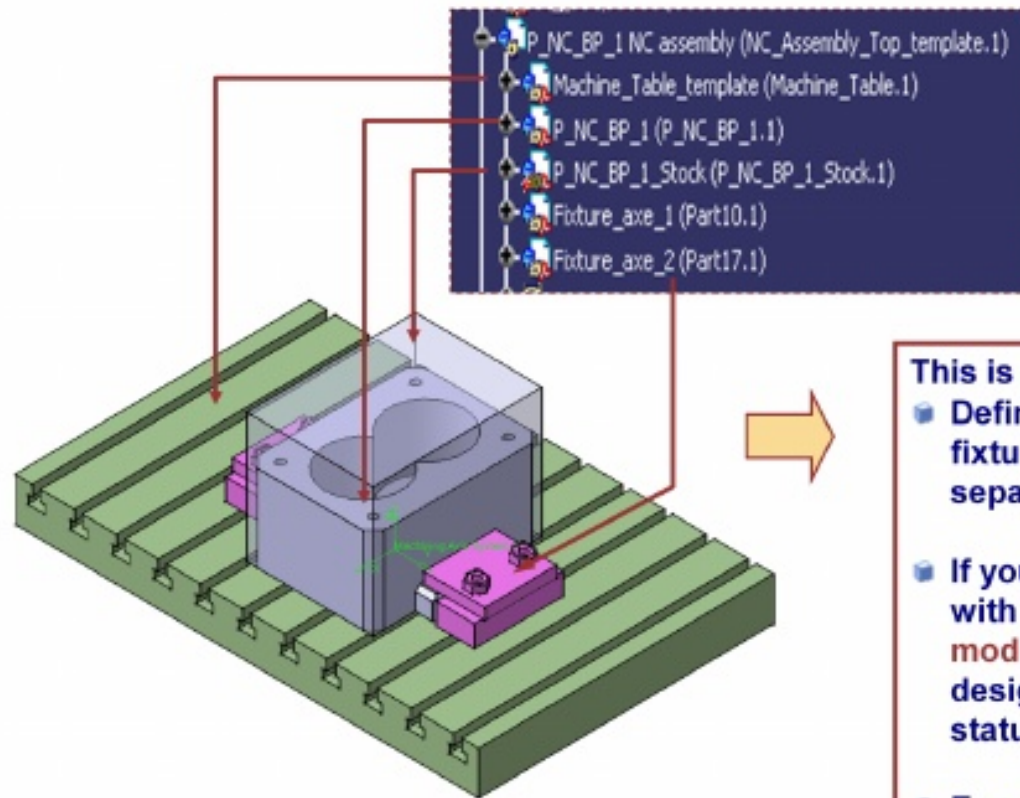# The Process Product Resources Model

The **Process Product Resources (PPR)** model is shared by all the Manufacturing applications (such as NC, Robotic, Welding, Painting, Inspection, etc) and can be accessed by a Process Planning Management tool

Process:
Part Operations

Machine set up

Manufacturing Program

Machining Operation

- P.P.R.
- ProcessList
  - Process
    - Part Operation.1
      - Manufacturing Program.1
        - Tool Change.1 T1 End Mill D 50 rc10
        - Roughing.1 (Computed)
          - Tool path
    - Machining Process List.1

> Process is the place where all the NC entities will be created by the user

Product:
Parts or Products used for Manufacturing: Design Parts, Fixtures, Stock, Manufacturing elements, etc

- ProductList
  - NCSetup_Part1_10.00.45 (NCSetup_Part1_10.0
  - Part1 (Part1.1)
  - NCGeometry_Part1_10.00.45 (NCGeometry_
    - NCGeometry_Part1_10.00.45
      - xy plane
      - yz plane
      - zx plane
      - PartBody
      - Geometrical Set.1

> Resources used in the Process are automatically listed in the Resources list and are available for the others Manufacturing applications and for a Process Planning Management tool

Resources:
Machines & Tools

- ResourcesList
  - 3-axis_Default_machine
  - T1 End Mill D 50 rc10

With Product and Resources Assignment, links are made and managed between the Design World (Product), the Manufacturing World (Process) and the Resources World.

Files Management

Before starting NC programming, it is better to create the NC Assembly and the good practice is to organize the product into specific parts.

P_NC_BP_1 NC assembly (NC_Assembly_Top_template.1)
Machine_Table_template (Machine_Table.1)
P_NC_BP_1 (P_NC_BP_1.1)
P_NC_BP_1_Stock (P_NC_BP_1_Stock.1)
Fixture_axe_1 (Part10.1)
Fixture_axe_2 (Part17.1)

This is useful for:

- **Define the Part, the stock, the fixture, NC added geometry in separated files/product**

- **If you want to share the design with several users and forbid any modification: just protect the design.CATPart with a read-only status**

- **Easy management of Hide/Show entities**

CATProcess Management (1/2)

1. The different geometries are separated in CATPart

2. The CATProduct links all the CATParts

3. The CATProcess contains the NC program & the resources
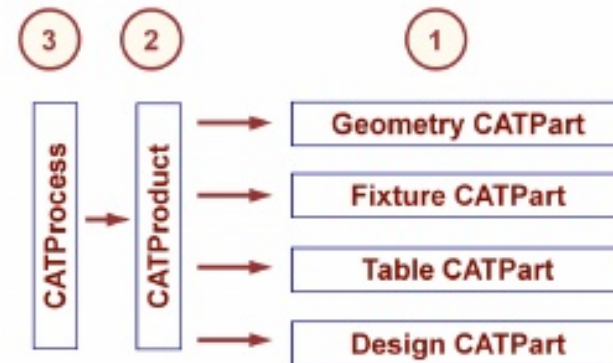
You will see how to modify this organization.

CATProcess Management (2/2)

What happens if you modify the Design.CATPart

In the PPR tree, update the CATProduct, then Update the CATProcess. See also « Design Associativity »
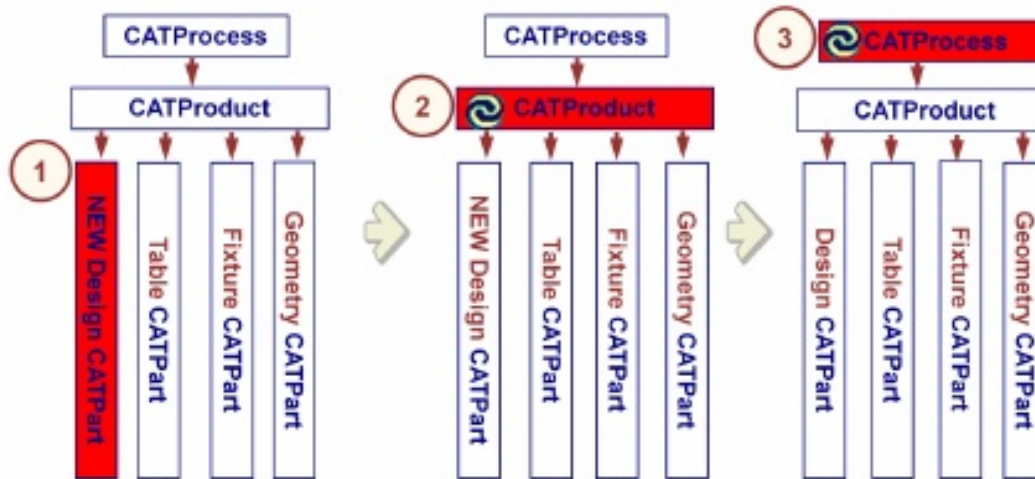
What happens if you move the Design.CATPart in your computer

Using the Links command (in the EDIT menu) update the link between the CATProduct and the CATPart.
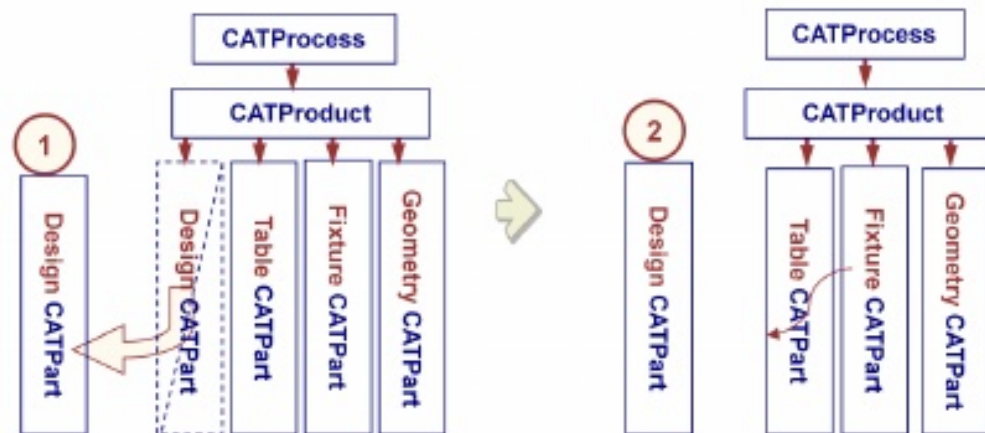
Status of the Machining Operations

All the Machining Operations displayed in the Manufacturing program may
have the following status:

	Operation computed (Tool path is computed)
	Operation Deactivate (done manually by the user)
	Operation Not Completed (Geometry is missing)
	Operation Not Updated (Tool path must be replayed to update the operation)
	Operation Locked (Machining Operation can't be modify)
	Tool path packed (Tool path is stored externally -on the hard disk-)

Other information is displayed between () at the end of the Machining operation name:

ZLevel.1 (Computed)   Operation computed (Tool path is computed)
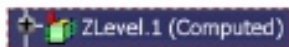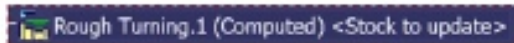
Rough Turning.1 (Computed) <Stock to update>   for Lathe operations with automatic stock option
up to date
to update

During the NC data output computation, if the system detects a Machining Operation with a
Deactivate or **Not Complete** Status, this operation is not taken into account in the computation.
In the resulting Report of the NC data output computation, a warning is generated to advise the user.

Student Notes:

Machine and Part Setup

This lesson covers the following topics:

- **About Setup**
- **Defining the Setup**
- **Part Operation**

About Setup

P.P.R.
- **ProcessList**
  - **Process**
    - Part Operation.1
      - Manufacturing Program.1
    - 3-Axis Roughing
      - Manufacturing Program.2
    - 5-Axis Finishing
      - Manufacturing Program.3
    - Machining Process List.1
- ProductList
- ResourcesList

The **Part Operation** is the NC entity that groups all the technological information necessary for part machining such as Machine-Tool, Set-up, Machining Axis System, etc.

Part Operation

Name: Part Operation.1
Comments: No Description

3-axis Machine.1

Default reference machining axis for Part Operation.1

Product3

| Geometry | Position | Simulation | Option | Collisions checking |

/Product3/NCAssemblySetup1/Aero_part/Aero_part/PartBody

/Product3/NCAssemblySetup1/Stock/Stock/PartBody

/Product3/NCAssemblySetup1/Fixtures/Fixtures/PartBody

1 safety plane selected

No traverse box plane selected

No transition plane selected

No rotary plane selected

OK   Cancel

Part Operation.1
- Roughing
  - Tool Change.1 T1 End Mill D 10
  - Tool Change.3 T3 End Mill D10
- Finishing
  - Tool Change.1 T4 Conical Mill D 25
  - Tool Change.2 T1 End Mill D 6

The **Manufacturing Program** describes the processing order of the NC entities that are taken into account for tool path computation: Machining Operations, Auxiliary Operations and PP Instructions.

- A Process tree can contain several Part Operations if it is necessary to change the machine-tool or the set-up for different machining phases
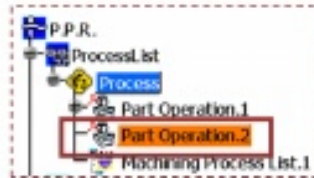- A Part Operation contains one or several Manufacturing Programs. These Manufacturing Programs can be organized by user choices:
    - Type of Activities (Roughing, Semi Finishing, Rework)
    - Tools (a program by tool)
    - Geometry (a program for all the pockets, for the holes)

Defining the Setup (1/3)

Define Name and Comment
Can be retrieved in the APT file

Define Machine Tool
- Machine type & axis
- Numerical Control options
- Compensation

Select type of machine or Import DELMIA Machine

Define outputs options

Define Compensation options

Define Default tool Catalog

Define Machining axis
- Spindle
- Rotable

Defining the Setup (2/3)

Define Machining axis System
- ◆ define Origin
- ◆ define X & Z axis

Select axis
- ◆ X & Z
- ◆ geometry selection or coordinates or angles

+

Select origin
- ◆ Coordinates or
- ◆ geometry (point)

OR

Select Plane
- ◆ select existing design axis system Or
- ◆ select existing Machining axis system

Part Operation

Name: Part Operation.1
Comments: No Description

3-axis Machine.1

Default reference machining axis for Part Operation.1

Product3

| Geometry | Position | Simulation | Option | Collisions checking |

//Product3/NCAssemblySetup1/Aero_part/Aero_part/PartBody
//Product3/NCAssemblySetup1/Stock/Stock/PartBody
//Product3/NCAssemblySetup1/Fixtures/Fixtures/PartBody
1 safety plane selected
No traverse box plane selected
No transition plane selected
No rotary plane selected

Default reference machining axis for Part Operation.1

Axis Name : Default reference machining axis for Part Operation.1

NC Output Parameters
☐ Origin   Number : 1   Group : 1

OK   Cancel

Define NC Assembly
- ◆ Yet defined if access to Machining workbench from NC assembly model
- ◆ select CATPart or CATProduct

Defining the Setup (3/3)

Part Operation

Name: Part Operation.1
Comments: No Description

3-axis Machine.1

Default reference machining axis for P.A Operation.1

A	B	C	D	E
Geometry	Position	Simulation	Option	Collisions checking

/Product3/NCAssemblySetup1/Aero_part/Aero_part/PartBody

/Product3/NCAssemblySetup1/Stock/Stock/PartBody

/Product3/NCAssemblySetup1/Fixtures/Fixtures/PartBody

1 safety plane selected

No traverse box plane selected

No transition plane selected

No rotary plane selected

OK   Cancel

A

Define geometry for Simulation
- Part
- Stock
- fixtures

Define Geometry for automatic creation of transition complete
- Rotation planes
- transition planes

B

Define Position options
- Tool Change point
- Table Center Setup

C

Define Simulation option
- Stock Accuracy

D

Define other options
- Intermediate stock for milling and turning operations
- Automatic stock selection for turning operations

E

Define Collision Checking option
- Activate Collision Checking option on design part or fixtures
- Offset on tool shank or tool assembly

Part Operation

You will learn how to insert and define a Part Operation and a Manufacturing Program in the process.

- **Need of a Part Operation**
- **Creating a Part Operation (PO): General Process**
- **Define the PO**
- **Need of a Manufacturing Program**
- **Multi Setup Management**
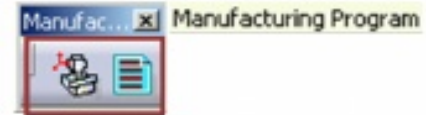
Why do you need a Part Operation

The Part Operation is the NC entity that groups all the technological information necessary for part machining such as Machine-tool, Set-up, Machining Axis System, etc.

Identification in tree

- P.P.R.
  - ProcessList
    - Process
      - Part Operation.1
        - Manufacturing Program.1
        - 3-Axis Roughing
          - Manufacturing Program.2
        - 5-Axis Finishing
          - Manufacturing Program.3
        - Machining Process List.1
  - ProductList
  - ResourcesList
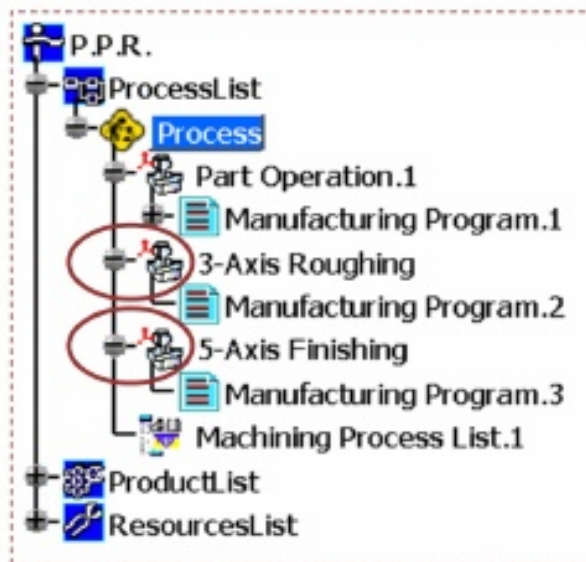
- A Part Operation references one machine tool

- A Part Operation defines a single part setup

- On a Part Operation, you can associate a Part or a Product to select geometrical elements

- The Machining Axis System is the default reference axis system for the coordinates of points generated in the APT or NC code.

A Process tree can contain several Part Operations if it is necessary to change the machine-tool or the set-up for different machining phases.

Creating a Part Operation: General Process

Click Part Operation Icon

The dialog box contains all the parameters necessary to define the new 'Part Operation'.

Type the Name & Comments. (optional because a default name is given by the system 'Part Operation.X')

The new Part Operation is created in the tree after the current one. Double-Click the Part Operation to edit it.

P.P.R.
ProcessList
Process
Part Operation.1
Part Operation.2
Machining Process List.1

Type the Part Operation specifications in the dialog box and click OK.

Part Operation

Name: Part Operation.1
Comments: No Description

3-axis Machine.1

Default reference machining axis for Part Operation.1

Product3

| Geometry | Position | Simulation | Option | Collisions checking |

/Product3/NCAssemblySetup1/Aero_part/Aero_part/PartBody
/Product3/NCAssemblySetup1/Stock/Stock/PartBody
/Product3/NCAssemblySetup1/Fixtures/Fixtures/PartBody
1 safety plane selected
No traverse box plane selected
No transition plane selected
No rotary plane selected

OK   Cancel

The comments will be generated at the start of the APT,CLFILE and NC code (optional) with the PPRINT prefix as all the comments available in the NC Entities like Machining Operations, PP Instructions, Machine-tool, etc.

Define the Part Operation: Defining a Machine-Tool (1/7)

Numerical Control:

Machine type: 3-axis w/ wo Table rotation, 5 Axis, horizontal/vertical Lathe, or multi turret machine

External Machine:
Select CATProduct from NC Machine tool builder
Or
Assign Machine from PPR Resources
for machine simulation

Post Processor / PP words table / NC data type:
PP / PP word table / NC data type taking by default when generating Outputs (APT, NC code ...)
NC Data Format you want to generated
(X,Y,Z or X,Y,Z,I,J,K)
Home Point Strategy * : what syntax you want to use for home point: From or Goto
Min/Max Interpolation radius * values used if you activate interpolation options
Min discretization step/angle values :
min distance / angle between two points for computation
Active Interpolation options *

* These values / selections are taking into account if you select « from machine » during the output creation

allows to output helical interpolation instructions for helix tool motion on Circular Milling and Thread Milling operations.
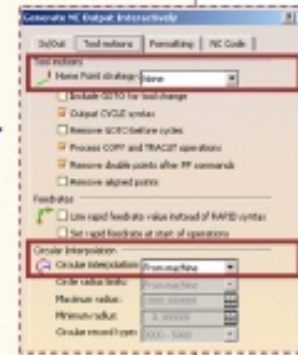
Define the Part Operation: Defining a Machine-Tool (2/7)
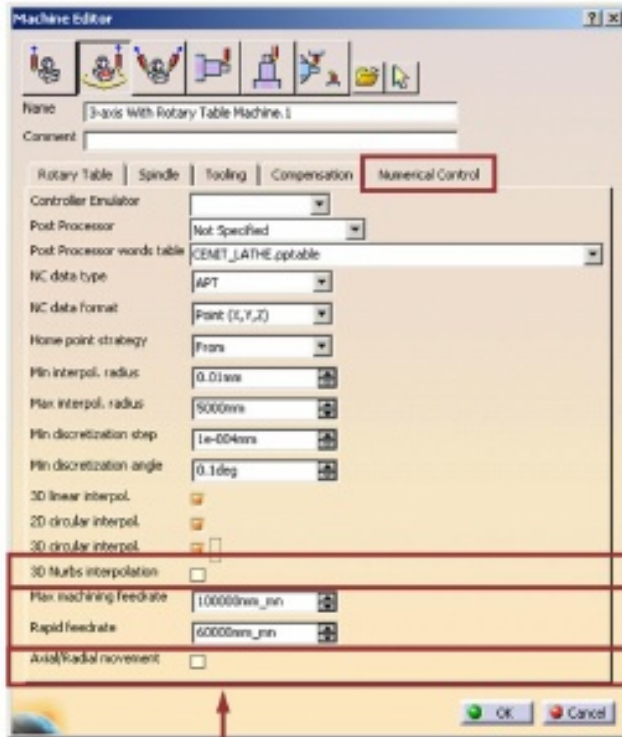
NURBS format output for Prismatic, 3-Axis and Multi-Axis Machining operations.
Reducing as much machining time and improving surface finish of machined parts of high-speed manufacturing processes
Only available for Siemens machines
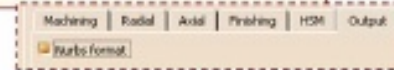
Max machining Feedrate: If you have define a Feedrate bigger than this value in a Machining operation,
- ◆ it will be detect as a collision in the report in Video simulation
- ◆ an error will be generated in the report file of the apt generation

Rapid feedrate * : value use to compute Rapid motion time, this value will be take into account if you decide to generate APT without RAPID instruction

* These values / selections are taking into account if you select « from machine » during the output creation
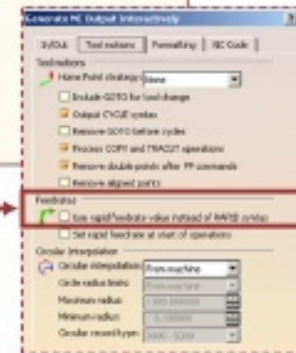
And define the mode of transition path between operation
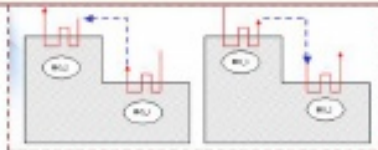
Define the Part Operation: Defining a Machine-Tool (3/7)

Compensation:
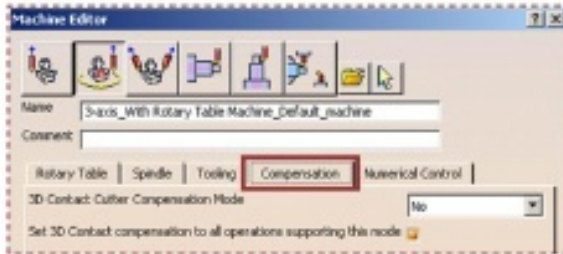
Activate 3D cutter compensation in contact mode or Tip & contact mode

| No |
| Contact |
| Tip & Contact |

Select the checkbox to impose 3D contact compensation to all operations supporting this mode whatever the choice defined at machining operation level

Summary of the different Compensation output modes available for each operation:
Cutter compensation instructions are generated on the NC data output depending on the selected mode.

Machining Operation	2D radial tip	2D radial profile	3D radial	3D contact
Profile Contouring (between planes)	Yes	Yes	-	-
Pocketing	Yes	Yes	-	-
Circular Milling	Yes	Yes	-	-
Sweeping	-	-	-	Yes
Between Contour	-	-	-	Yes
Spiral Milling	-	-	-	Yes
Z Level	-	-	-	Yes
Sweep Roughing	-	-	-	Yes
Face Isoparametrics	-	-	-	Yes
Multi Axis Sweeping	-	-	-	Yes
Multi Axis Curve (Contact)	-	-	-	Yes
Multi Axis Contour Driven	-	-	-	Yes
Multi Axis Helix	-	-	-	Yes
Multi Axis Flank Contouring	-	Yes	Yes	-

Define the Part Operation: Defining a Machine-Tool (4/7)

Tooling:
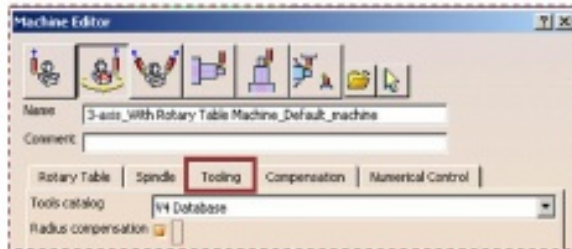
Tool catalog which is selected by default when you create a new operation or make a query.

Radius Compensation availability:
Activate the possibility to put a Radius compensation number on the tool that will be generated in the output.
(a new parameter is added in the compensation tab page in the tool definition panel)
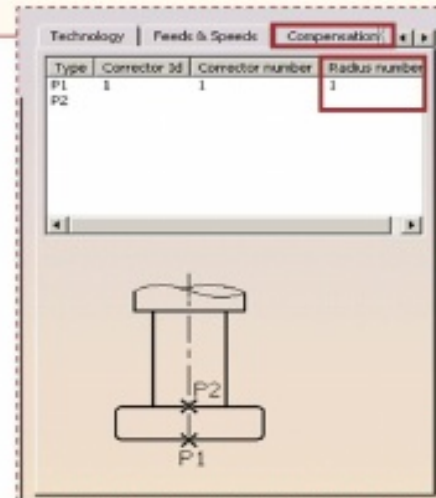
Define the Part Operation: Defining a Machine-Tool (5/7)

Spindle Data:

3-axis Machine with/ without rotary table
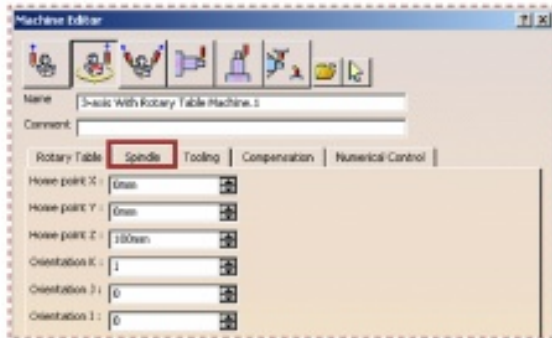
Spindle Data:
- ◆ Coordinates of the Home point
- ◆ Initial Axis orientation of the spindle

Lathe Machine

Multi Axis Lathe Machine

Spindle Data:
- ◆ Create spindles
- ◆ Spindle name & number
- ◆ Spindle Axis system
  - • origin point
  - • radial & axial axis
- ◆ Spindle max feedrate

See dedicated training for more information

Spindle Data:
- ◆ Spindle and radial axis
- ◆ Coordinates of the Center point
- ◆ Initial angular position
- ◆ Rotary angle
- ◆ Rotary Direction
- ◆ Rotary Type

All the coordinates are given according the Reference Machining Axis System.
These parameters are set automatically if you have associated a DELMIA Machine.

Define the Part Operation: Defining a Machine-Tool (6/7)

Rotary Table (3 axis machine with Rotable):

Allows ROTABL/ Output between points for Axial operation
Need to activate the option also in the axial operation

> Rotary motions are displayed during Replay. ROTABL/ Instructions are generated in the Output File.
> Facilitates environment set up by minimizing the requirement on post processors (not having to deal with X, Y, Z, I, J, K output in case of rotary table).
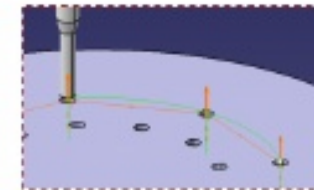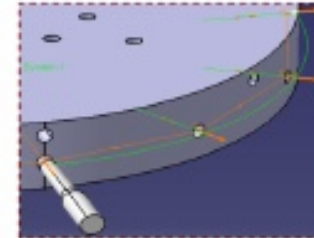
Rotary Table Data:
- **Coordinates of the table Center point**
  offset between physical Machine rotable center and Part Operation Machining axis System origin (value taking into account to calculate rotable matrix)
- **Initial angular position: position of the table before the first NC operation**
- **Rotary angle: minimum angle necessary to generate Rotable instruction**
- **Rotable Axis, Rotary Direction & Rotary Type: option use for Rotable instruction**

**All the coordinates are given according the Reference Machining Axis System. These parameters are set automatically if you have associated a DELMIA Machine.**

Define the Part Operation: Defining a Machine-Tool (7/7)

Turret:

Lathe Machine

Turret Data:
- Coordinates of the home point
- Initial Axis orientation

Multi axis Lathe Machine

Turret Data:
- Create turret
- Turret name & number
- Turret Axis system
  - origin point
  - radial & axial axis
- Tool change point coordinates

See dedicated training for more information

TRAVERSER (Vertical Lathe Machine)

Traverse Data:
- Number
- pitch

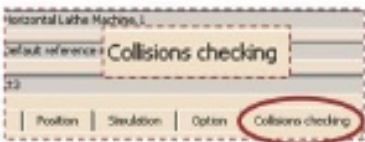All the coordinates are given according the Reference Machining Axis System.
These parameters are set automatically if you have associated a DELMIA Machine.

Define the Part Operation: Creating the Reference Machining Axis
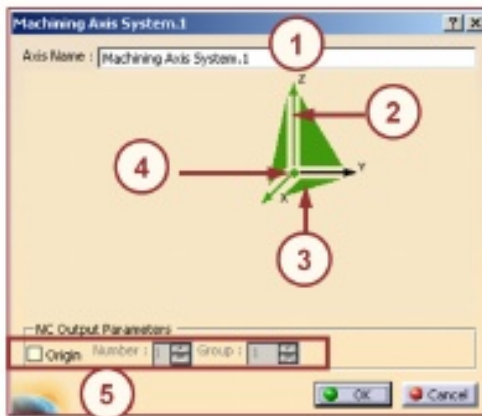
- **Name of the Reference Machining Axis System displayed in window (1)**

- **Select the arrows to define Z and X axis (2):**

  Axis Direction:
  - **Selection: Selecting an element (Line, Fsur, Edge on the part)**
  - **Manual:**
    - **By coordinate: X,Y,Z**
    - **By angle: select reference axis and define the rotary angles**
  - **Point in the view: select two points to define the orientation**

- **Select the planes to select an already existing Machining or design Axis system (3)**

- **Select point to define the origin of the machining axis system (4):**
  - **On the design**
  - **Using MB3 and key coordinates**

- **Origin check box (5):**
  Activate the Origin and specify the Number and Group to generate the following syntax in the Apt Source: ORIGIN/ X, Y, Z, Number, Group

All the output coordinates generated in the Apt, CLFile or NC Code are computed according the current Machining Axis System.

Define the Part Operation: Associating a Product or a Part to a Part peration

You can associate different products for each Part Operation to manage the Part Positioning and specific Set-Up.
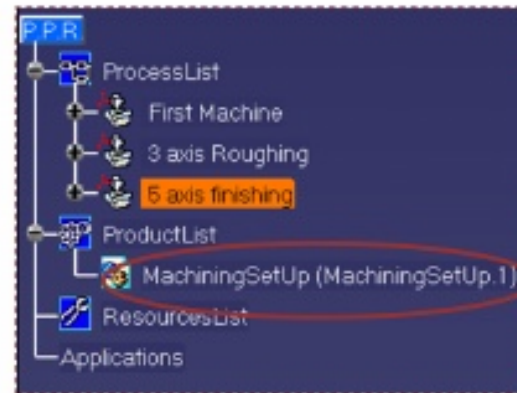
Select a Product or a Part to link this element to the Part Operation. This Product or Part is automatically referenced in the Product List in the PPR tree associated to the CATProcess

The Product can contain several parts:
- **The Design Part**
- **The Stock**
- **The Fixtures**
- **The Manufacturing Geometrical Data**
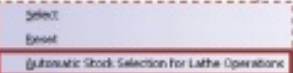
A Product can be automatically added to store NC geometry if you select the option in Tool/Option/Machining setting

Define the Part Operation: Define Geometrical Element for Computation and Simulation
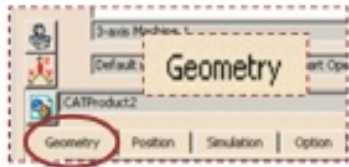
A. Define the part:
Allows you to make analyze in the Simulation dialog box see Lesson 4

B. Define the Stock:
- Use for Material Removal simulation
- Use for Computation to make Rework Roughing in SMG (don't define the stock on the SMG Roughing Operation)
- Use for automatic stock computation for LMG operation (need to activate the option : right-click on the stock field and select

> Select
> Reset
> Automatic Stock Selection for Lathe Operations

C. Define Fixture:
Allows you to visualize them in the Material removal simulation and detect collisions

D. Safety plane:
Default plane used if there is a motion to/from plane define in the macro motions (approach, retract,)
If you use auto-complete functionality, define instead transition, traverse box, rotary plane)

E. Traverse box plane:
Select 5 planes that define a global traverse box for the part operation.

F. Transition plane:
select the required planes that will be used as a global transition planes for the part operation when using auto complete functionality.

G. Rotation plane:
Select the required planes that will be used as a global rotary planes for the part operation when using auto complete functionality

Notice: Offsets can be added on all planes using right-click on the plane on the 3D model

Define the Part Operation: Defining the Machine Position Setup
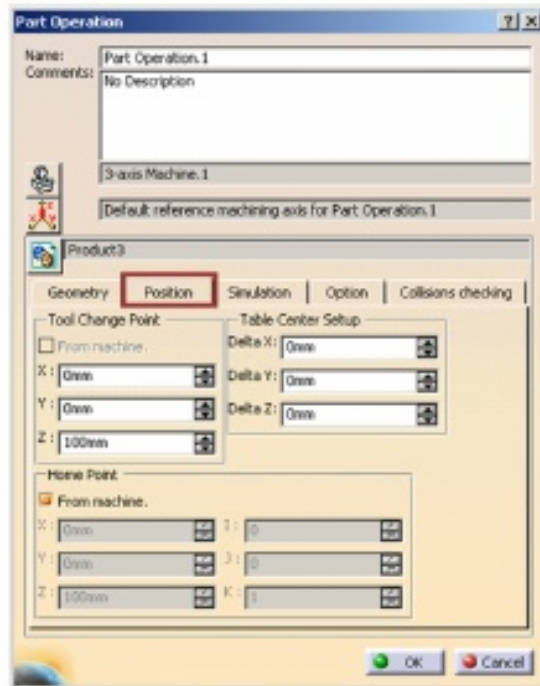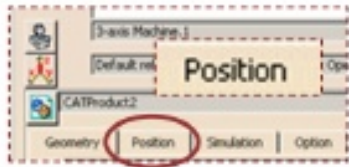
Table Center Setup:
This translation is used to fix the physical origin of the machine table according the Part Operation Reference Machining Axis System
These values are added to the one you define at the machine level.

Tool Change Point:
Define the tool change point.
For DELMIA machines change point is read from the machine and cannot be modified in the Part Operation.
For Multi-axis lathe machines, the tool change point is read from the machine and cannot be modified in the Part Operation.

Home Point:
Position of the different axis at the home position
If a DELMIA machine is already associated, these values can be set automatically from it.

All the coordinates are given according to the Reference Machining Axis System.

Define the Part Operation: Defining Simulation and Other Options
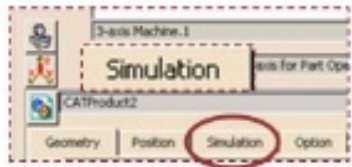
Define the value of the stock tessellation tolerance for the simulation
Default value = 0.2 mm
Can be decrease to µm
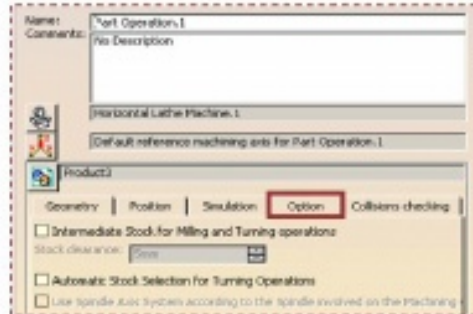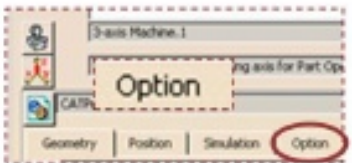
Intermediate stock for Milling and Turning operations
Activate the automatic stock computation to avoid stock selections in turning operations
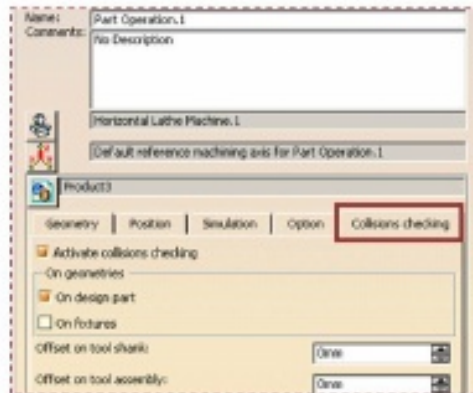
Activate the Collision Checking check box to detect any collision between the Design part and the tool

All the coordinates are given according to the Reference Machining Axis System.

Why Do You Need a Manufacturing Program

A **Manufacturing Program** describes the processing order of the NC entities that are taken into account for tool path computation: Machining Operations, Auxiliary Operations and PP Instructions.

P.P.R.
ProcessList
  Process
    Part Operation.1
      Manufacturing Program.1
        Tool Change.2 T1 Face Mill D 50
          Facing.1
        Tool Change.3 T1 End Mill D 10
          Pocketing.1
      Manufacturing Program.2
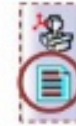
- The screen display is done by tools.
- Operations Management (Create, Edit, Copy, Move,etc) is available in a Manufacturing Program or between Manufacturing Programs.
- Tool path simulation is done operation by operation.
- Automatic generation of Tool Change and Table Rotation orders is done at Manufacturing Program level.
- NC output data can be generated at Manufacturing Program level.

The new Manufacturing Program is created after the current entity (Part Operation or Manufacturing Program)

A Part Operation contains one or several Manufacturing Programs.
These Manufacturing Programs can be organized by user choices:
- Type of Activities (Roughing, Semi Finishing, Rework,etc)
- Tools (a program by tool)
- Geometry (a program for all the pockets, for the holes,etc)

Part Operation.1
  Roughing
    Tool Change.1 T1 Face Mill D 50
    Tool Change.2 T1 End Mill D 10
  Finishing
    Tool Change.3 T3 Conical Mill D 25
    Tool Change.4 T4 End Mill D 6
  Drilling
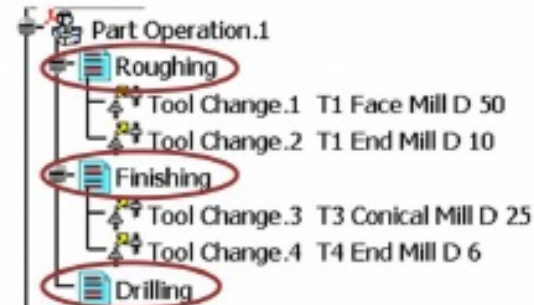
What is Multi Setup Management

In the Manufacturing Process, each time you need to change the positioning of the part on the machine, you need to create a new Part Operation.
It is better to create 1 CATProduct for each setup. Then associate each product to the good Part Operation.
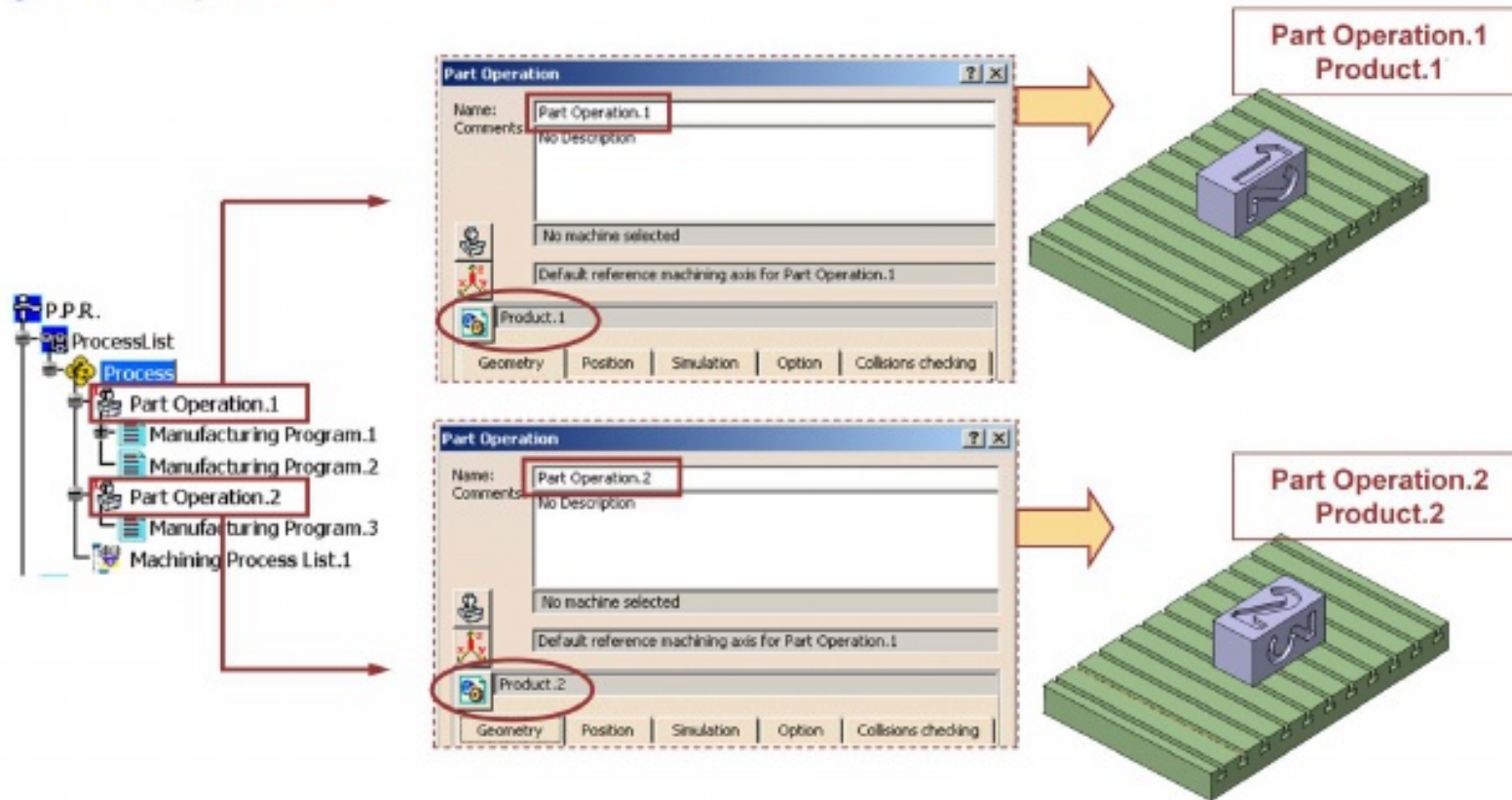
About the Intermediate Stock

You can compute & visualize the input and output intermediate stock for all types of machining operations. This intermediate stock helps you to optimize and compute a collision free toolpath.

The input stock of an operation is the output stock of the previous operation.

You can mix different types of operations in your program irrespective of the sequence of those operations (milling & turning).

For machining operations, there are four options for managing the intermediate stocks:

- ⬤ **Input Intermediate Stock:** Solid corresponding to the machined part before the machining operation.
- ⬤ **Output Intermediate Stock:** Solid corresponding to the machined part after the machining operation.
- ⬤ **Ignore Input Stock:** the computation of the tool path of the selected operation does not take into account of the intermediate stock input.
- ⬤ **Desactivate Output Stock Computation:** it deactivates the computation of the output intermediate stock for the selected operationAlso the selected operation and the following ones have the same intermediate stock as input.

Activate the checkbox in Part Operation > Option tab to compute the intermediate stock
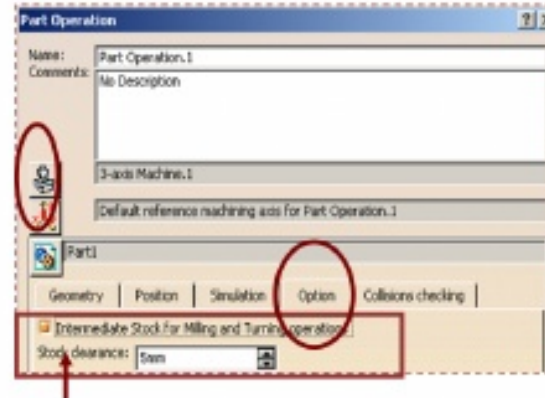
The Stock clearance is the safety distance on the intermediate stock

These 2 fonctions are also available with contextual menu

> 😊 **The intermediate stock bodies are not stored. They are available only during the session of the CATProcess.**

How to Use the Intermediate Stock

You will see how to compute and visualize the input and output intermediate stock in a program.

1. In Part Operation, define design part & stock
2. Select the checkbox for Intermediate stock. Enter stock clearance as 2 mm.
3. Create any machining operation. The input intermediate stock is displayed in 3D viewer.
4. Define the operation parameters and replay the tool path.
5. Click on Update Output Stock. The output stock is computed and displayed.
6. Create next operation and define its parameters.
7. Update the Input and Output stocks.
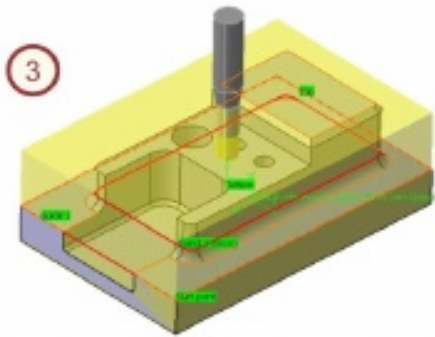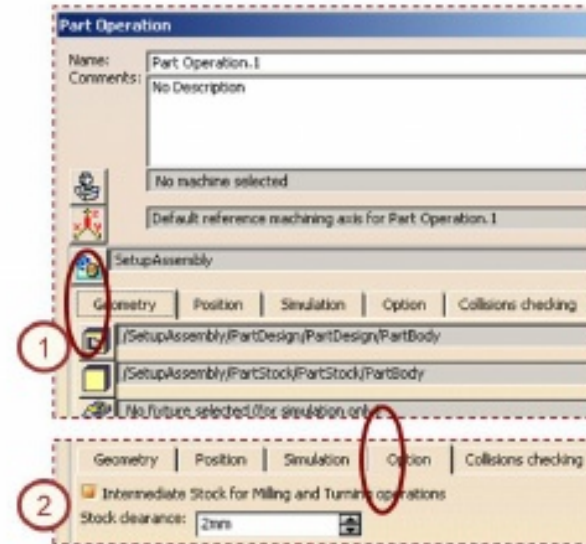
Input Intermediate Stock
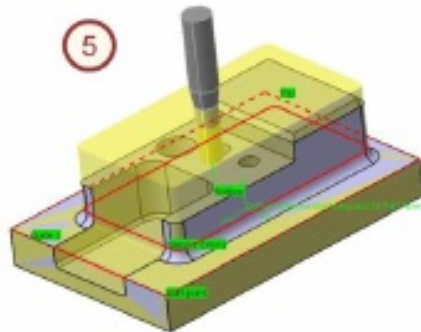
Output Intermediate Stock

Student Notes:

Machining Operation Definition

This lesson covers the following topics:

- Machining Operations Presentation
- Tools and Tool Assembly
- Feedrates Computation
- Macro Motions
- Axial Operations
- Process Views
- Geometry Wizard (Edge, Face)
- Tool Path Verification and Simulation

Machining Operation Presentation

You will see the Machining Operation Presentation.

- Introduction
- General Process
- Strategy
- Geometry
- Tool Assembly
- Feedrates
- Macro Motions

Machining Operations Presentation: Introduction

In CATIA V5 we can create machining operation from 2 to 5 axis

- Turning operations
- Milling operations
- Drilling operations

- Roughing operations
- Finishing operations

All the operations are defined in the same way

Machining Operations Presentation: General Process
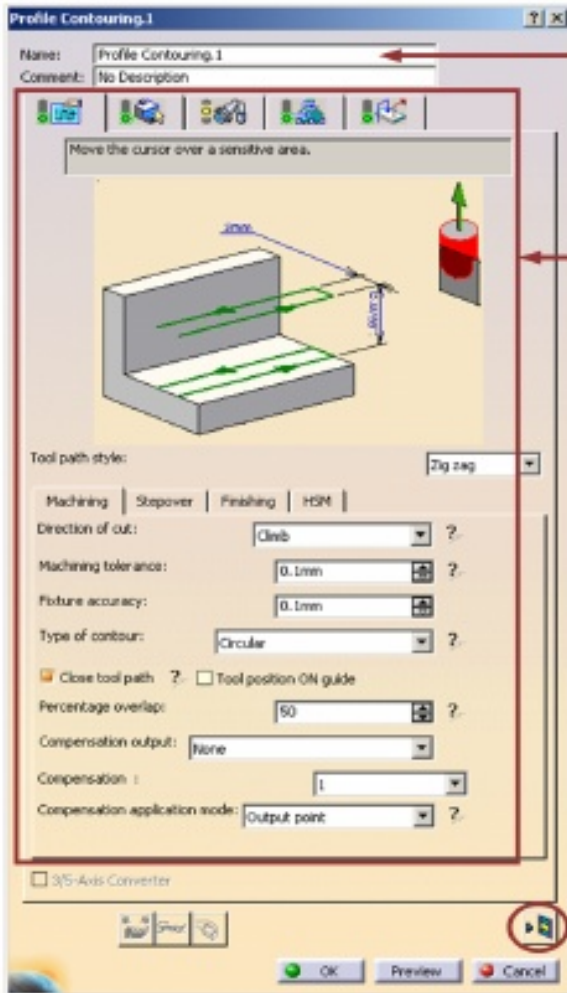
Name of the Operation and Comment

Define machining operation parameters concerning:

Strategy   Geometry   Tool   **Feeds & Speeds ▼**   Macros

You can compute the tool path only if the light are all green or orange
- ◆ **Green: geometry selected**
- ◆ **Orange: geometry optional**
- ◆ **Red: missing geometry**

For each Tab page, define your parameters, click on « ? » to have help
A picture shows you, the strategy used
　　Double-click the blue value to modify it

Replay and / or Simulate the operation tool path

Machining Operations Presentation: Strategy
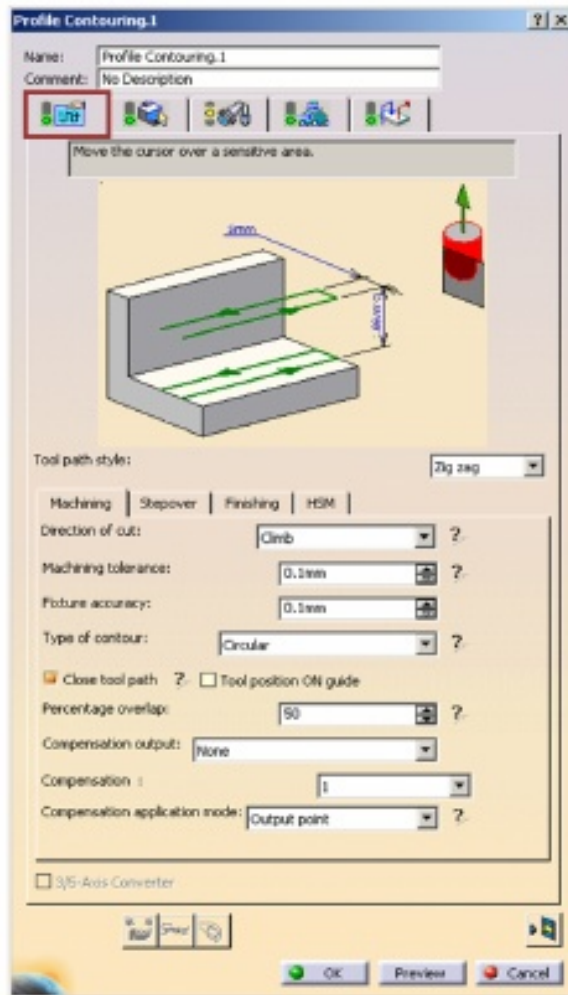
Strategy Tab:

Define tool path style

Define Machining parameters:
- ◆ **Direction of cut**
- ◆ **Machining Tolerance**

Define Stepover (Axial & Radial) parameters
- ◆ **Number of levels**
- ◆ **Max depth of cut**
- ◆ **Scallop height**

Define Finishing parameters
if you want to include a finishing/semi finishing path on the bottom/side

Define High Speed milling parameters corner radius

Machining Operations Presentation: Geometry
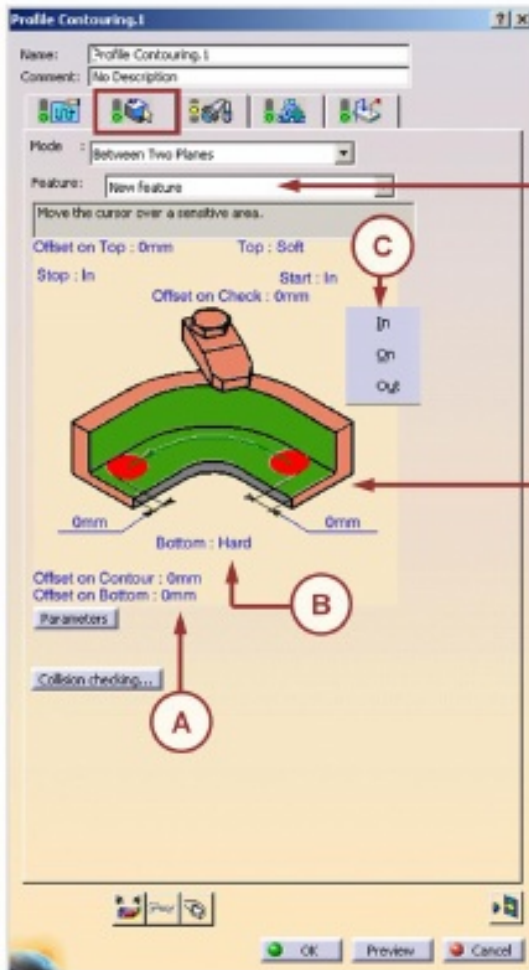
Geometry Tab:

You can select a predefined Machining area or define manually all the elements:

Define Geometrical Elements
- **Part / Stock / Check**
- **Top / Bottom / Imposed plane**
- **Limiting element**

Code color:
- **Green: geometry selected**
- **Orange: geometry optional**
- **Red: missing geometry**

Define Offset on geometrical elements
- **Double-click the value to modify it (A)**
- **For parameters with only 2 possible value: click on the parameter to invert the selection (B)**
- **For parameters with more than 2 possible values:**
  MB3 and select the good value (C)

Machining Operations Presentation: Tool Assembly

Select From Catalog or External Database:
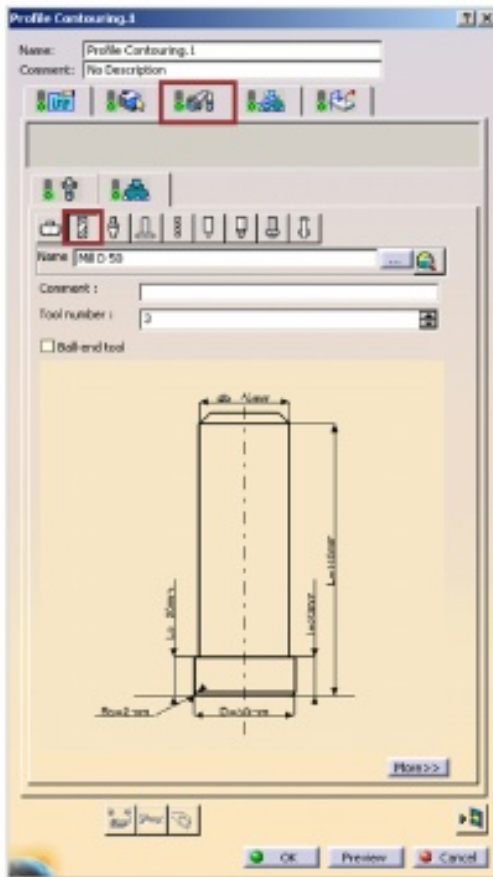
Tool Assembly query:

- ◆ Select the tool catalog or the external database (1)
- ◆ Select the type of tool (2)
- ◆ Make queries (optional) (3)

You can select a tool assembly (holder + tool) by selecting a tool (bottom window) or a assembly (top window)
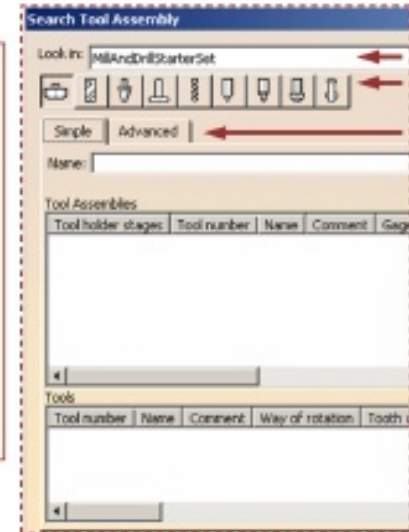
Tool query:

- ◆ Select the tool catalog (1)
- ◆ Select the type of tool (2)
- ◆ Make queries (optional) (3)
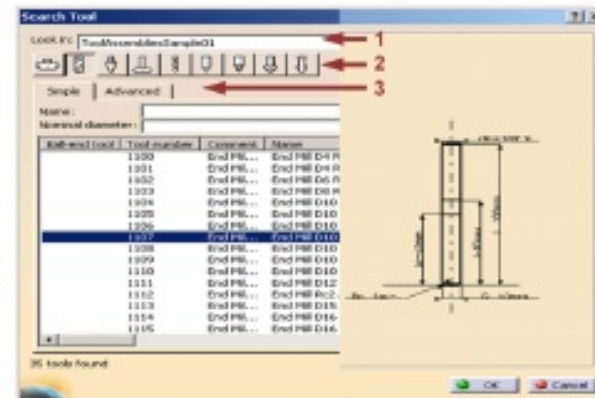
Machining Operations Presentation: Feedrates
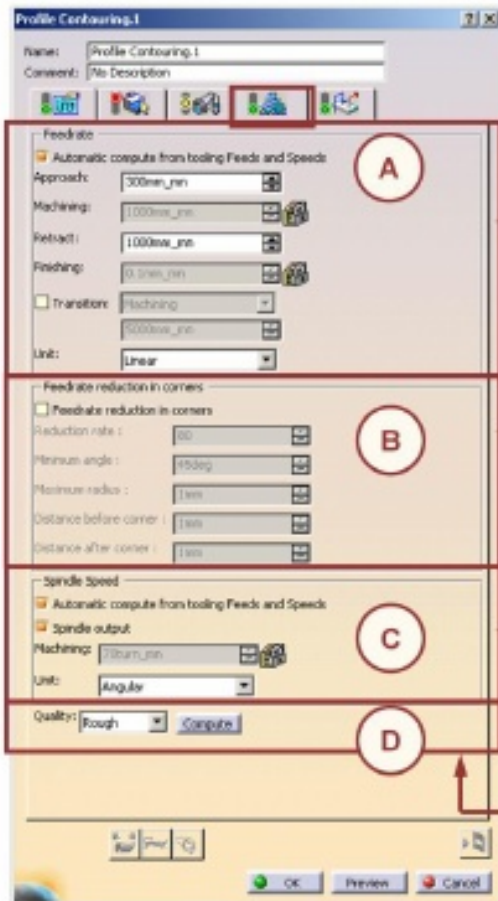
A- Feedrate definition:
- Activate « Automatic compute » if you want to compute feedrate from tool values
- Deactivate « Automatic compute » to type your own values

B- Definition of Feedrate reductions in corners
- Activate the option to reduce Machining speed in the corner

C- Spindle speed:
- Activate « Automatic compute » if you want to compute feedrate from tool values
- Deactivate « automatic compute » if you want to type your own values

D- Define quality: (for automatic computation from tool parameters)
- Select Rough or Finish to load on the tool the appropriate set of feeds & speeds values
- Click compute to update the values in the operation feedrate tab page

Machining Operations Presentation: Macro Motions
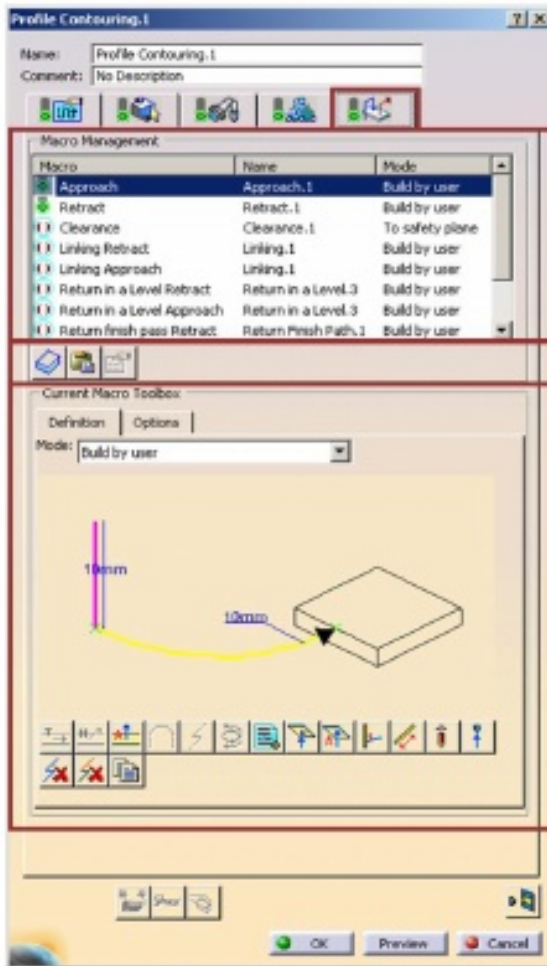
Macro Motions:
Macro motion allows to define in the machining operation parameters for approach, retract & linking motions.

Macro selection:
   ◆ Select the macro you want to define
   ◆ Activate / inactivate Macro motions (right-click)
A symbol indicates you the status of the macro (deactivate, geometry is missing, OK)

Store macro in your **catalog** or **retrieve** macro from catalog

Macro definition: Definition Options
   ◆ Select the type of macro (predefined or not)
   ◆ Build it or change parameters (double-click or contextual menu)
   ◆ Affect geometry
   ◆ Insert PP word (MB3 on green cross)

Macro option: Definition Options
   ◆ Name the macro
   ◆ Key a comment
   ◆ Activate or not « cornerized clearance with radius »

Tools and Tool Assembly

You will see how to import, select and create Tools and Tool Assembly.

- **Tool Tab Presentation**
- **Selecting a Existing Tool or Tool Assembly**
- **Importing Tools in the Resources List**
- **Creating a Tool Catalog from the Resource List**
- **Creating a Tool or Tool Assembly**

Tools Tab Presentation

1. **Select the Assembly or the tool to be defined.**
   If you select an assembly, the associated tool is set
   automatically but you have the possibility to change it.

2. **Select the kind of tool you want**

3. **Access to tool query windows**
   - Select a tool already used in the document
   - Select a tool in a catalog or in a external database

4. **Define a comment and the tool number if necessary**

5. **You can use 2D viewer for editing tool characteristics**
   by double-click on the values and access to more
   parameter by clicking on   More>>
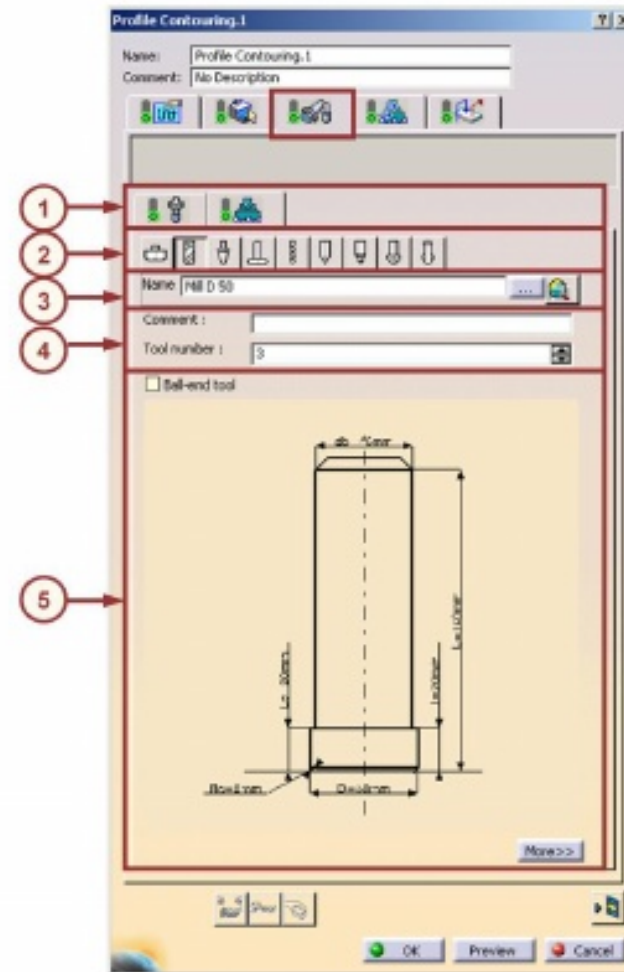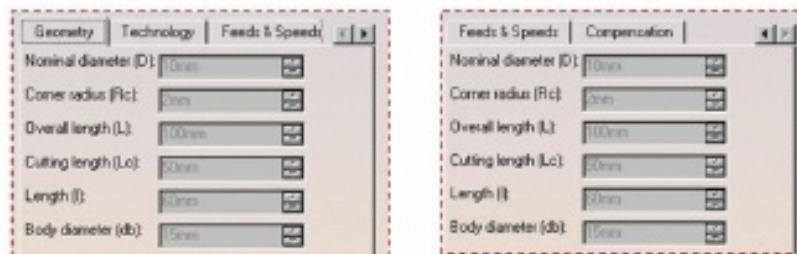
Selecting a Existing Tool or Tool Assembly
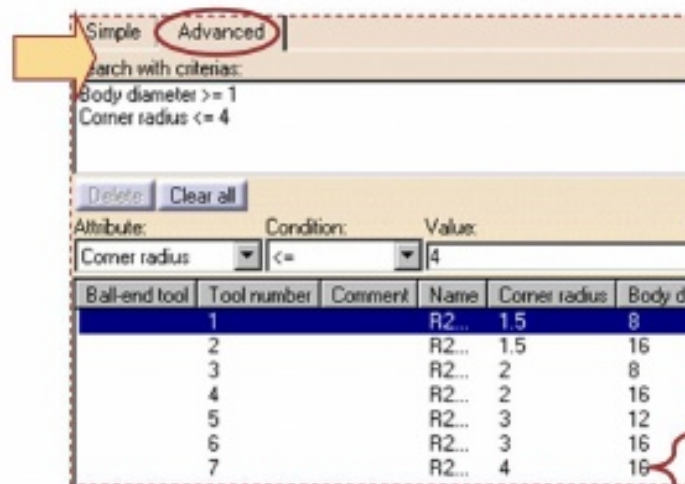
Tool Queries:

Simple query: From a Tool type List proposed with the current Manufacturing Operation, select a particular tool type or all Tools with 🔧
Specify:
- ◆ The name of the tool (or a part of it) and/or
- ◆ The tool diameter
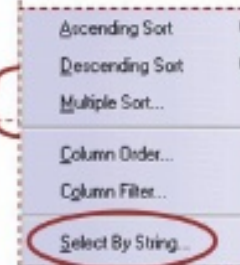
Advanced query: create criteria for search via Attribute-condition-value settings
To appear in the result list, a tool must meet all criteria

Via **contextual menu** you can:
- ◆ **Reorder the list of attributes**
- ◆ **Look for a tool by a character string**

Ball-end tool	Tool number	Comment	Name	Corner radius	Body dia
	1		R2...	1.5	8
	2		R2...	1.5	16
	3		R2...	2	8
	4		R2...	2	16
	5		R2...	3	12
	6		R2...	3	16
	7		R2...	4	16

Ascending Sort
Descending Sort
Multiple Sort...
Column Order...
Column Filter...
Select By String...

> 😎 **You can assign/ replace the tool assembly on a set of multiple Machining Operations from contextual menu.**

Importing Tools/Tool Assemblies in the Resources List

You can import existing Tools and Tool Assemblies from a Catalog or Database.
In this case there is no need to have an existing machining operation.

1. Click Import/List Tools or Tool Assemblies icon
   The Search Tool dialog box is displayed
2. Select tools catalog via Look in item
3. Select the type of search you want
4. You can make queries on tool parameters if necessary
5. Select your tools in the list
   The selected tools or tool assemblies are automatically added in the Resources List and available now for queries in the document.

You can import one or more Tool Assemblies for a given tool type from catalog or database.
The Search Tool Assembly functionality is almost the same as for selecting a tool assembly by means of a query with query icon.
The only difference is the list of tool types that shows all the tool types according to the active Machining workbench.
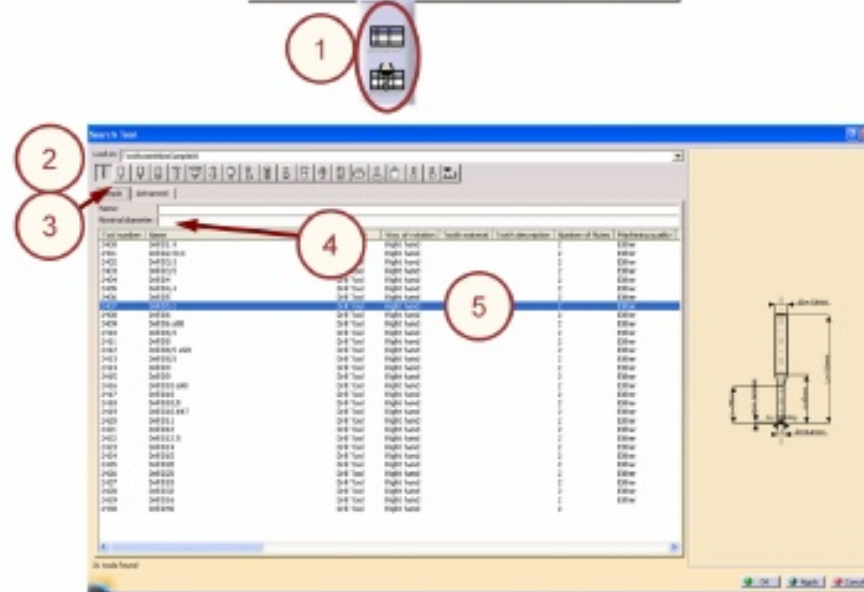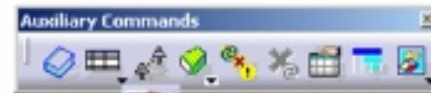
Creating a Tool Catalog from the Resource List

You can create a tool catalog from selected tools in the resource list. A tool catalog can contain tool assemblies, tools and tool inserts.

1. Select the tool/s in the resource list (shift/ctrl key for multi selection)

2. Right-click and select Send to Catalog The Save in Catalog dialog box is displayed that allows you to create a new tool catalog or overwrite an existing one.

3. To create a new tool catalog, click the [...] button to navigate to the required folder and type a new name for the catalog. Otherwise save with the name of the required catalog to overwrite.

4. Click OK to create the new or updated catalog. The resulting tool catalog appears in a new Catalog Editor window

Importing Tools/Tool Assemblies in the Resources List

You can import existing Tools and Tool Assemblies from a Catalog or Database.
In this case there is no need to have an existing machining operation.

1. Click Import/List Tools or Tool Assemblies icon
   The Search Tool dialog box is displayed
2. Select tools catalog via Look in item
3. Select the type of search you want
4. You can make queries on tool parameters if necessary
5. Select your tools in the list
   The selected tools or tool assemblies are automatically added in the Resources List and available now for queries in the document.

You can import one or more Tool Assemblies for a given tool type from catalog or database.
The Search Tool Assembly functionality is almost the same as for selecting a tool assembly by means of a query with query icon.
The only difference is the list of tool types that shows all the tool types according to the active Machining workbench.

Creating a Tool or Tool Assembly (1/5)

Define Tool Assembly:

Define Holder:

1. **Access to the tool definition Panel**

2. **Select Assembly tab page**

3. **Create a new holder:**
   - ◆ Key the name of the assembly
   - ◆ Define Comment, tool number, number of stages & power
   - ◆ Define assembly parameters (geometry, technology)
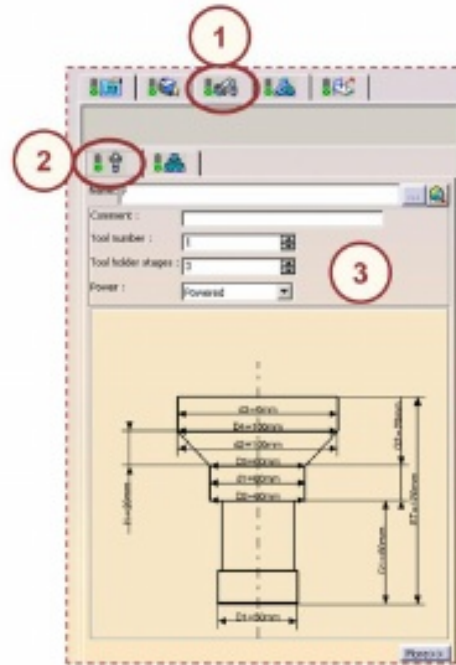
Define Tool:

4. **Select tool tab page**

5. **Select tool from document or from catalog or**

6. **Define a new tool**
   - ◆ Key the name of the tool
   - ◆ Define Comment, tool number
   - ◆ Define assembly parameters (geometry, technology, feeds & speeds, compensation)
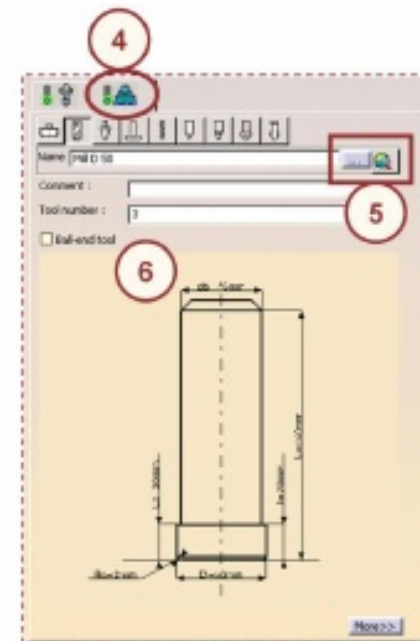
Creating a Tool or Tool Assembly (2/5)

Details of Holder Parameters:

- **Name**
- **Tool Number: value generated in outputs**
- **Number of stages:**
    1. a stage can be cylindrical or conical (3 parameters to define it: length, diameter1, diameter2)
    2. number max of stage = 5
- **Power: fixed for turning tool, powered for milling tool**
- **Geometrical parameters: double-click the value to modify it**

More>> **Access to the full geometrical and technological parameters**

Geometry tab page:
D1: tool diameters
ST: total length (tool + holder)
Set X,Y,Z :
Orientation: tool assembly setup angle
Dx, dx: Diameters of the different stages
Lx: length of the different stages

Technology tab page:
Number of components
Gx: value of the gages
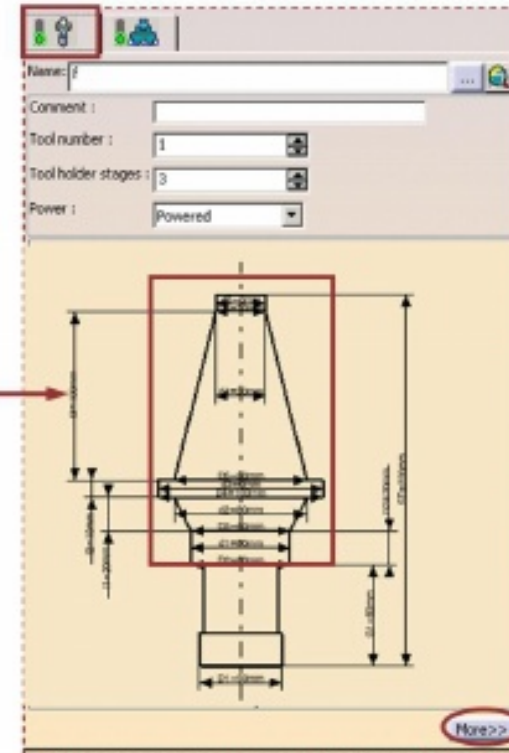
Creating a Tool or Tool Assembly (3/5)

Details of Tool Parameters: (1/3)

- Tool type
- Name
- Tool Number: value generated in outputs if no assembly has been defined
- Comment
- Ball end: activate it then all corner radius are equal to tool nominal radius

- Geometrical parameters: double-click the value to modify it

More>> Access to the full geometrical and technological parameters

Geometry	Technology	Feeds & Speeds	C
Nominal diameter (D):	50mm		
Corner radius (Rc):	0.8mm		
Overall length (L):	150mm		
Cutting length (Lc):	19mm		
Length (l):	19mm		
Body diameter (db):	45mm		
Non cutting diameter (Dnc):	0mm		
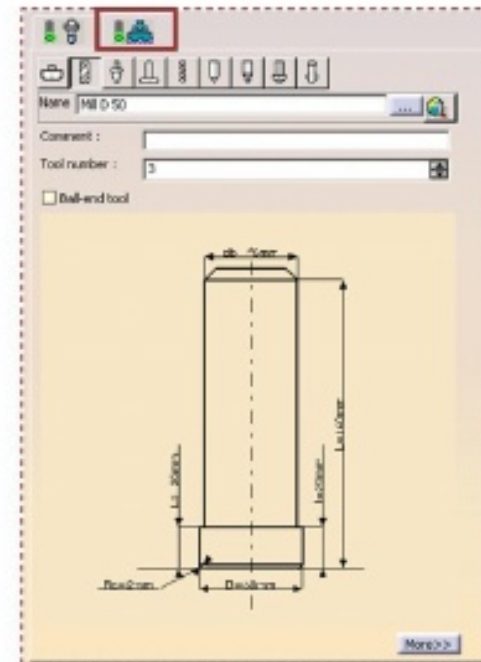
See 2 Next foils for the detail of each tab page

Creating a Tool or Tool Assembly (4/5)

Details of Tool Parameters: (2/3)

Geometry parameters:

D : Cutting diameter

L : tool total length

Lc : tool cutting length

Db : body diameter

Dnc : inner non- cutting diameter

Technology parameters:

All theses value can be used to make formula

All these values (or combination of theses values) can be retrieve in the APT / NC code

Creating a Tool or Tool Assembly (5/5)

Details of Tool Parameters: (3/3)
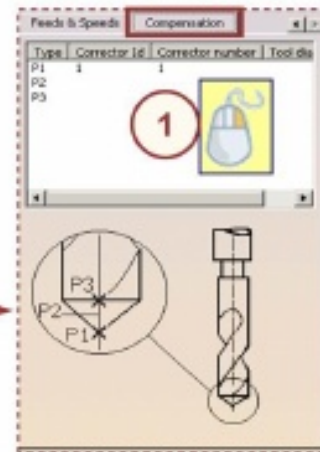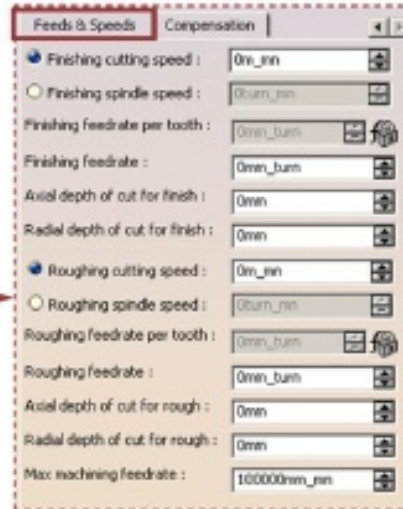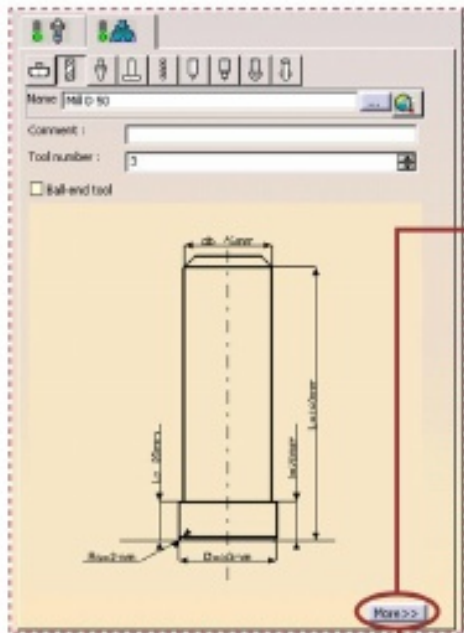
Feeds and Speeds:

Optimal Feeds and Speeds and cutting depth values recommended by the tool maker.

Only feeds and speed values can be automatically used by the system (for time computation).

Compensation:

1. **MB3 on Compensation site to edit**
2. **Modify the Compensation parameters**
   - **Corrector Number**
   - **Length Register Number**
   - **Radius Register Number**
   - **Tool Diameter to specify the compensation site location (for example: the site P2 of the drill)**
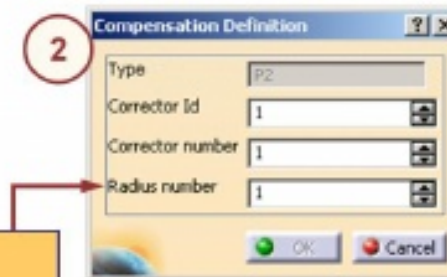
The Radius register number is only available if the option Radius Compensation has been activated on the Machine-tool.

Creating a Tool/Tool Assembly from Scratch

You can create tool and tool assembly from scratch using the Tool change command directly on the Resource List node without creating any machining activity.

1. Select the Resource List node in the specification tree.

2. Click any Tool Change.
   The Tooling Creation dialog box displays for defining the tool or tool assembly.

3. You can change tool parameters of the default tool

4. Click Assembly tab.
   The empty page with Name field displays. Type the name of the assembly that to be created. The tool assembly along with the tool defined in the tool tab is displayed. You can change the geometrical and technological parameters of the tool assembly.

5. Click OK to create the tooling in the Resource List.
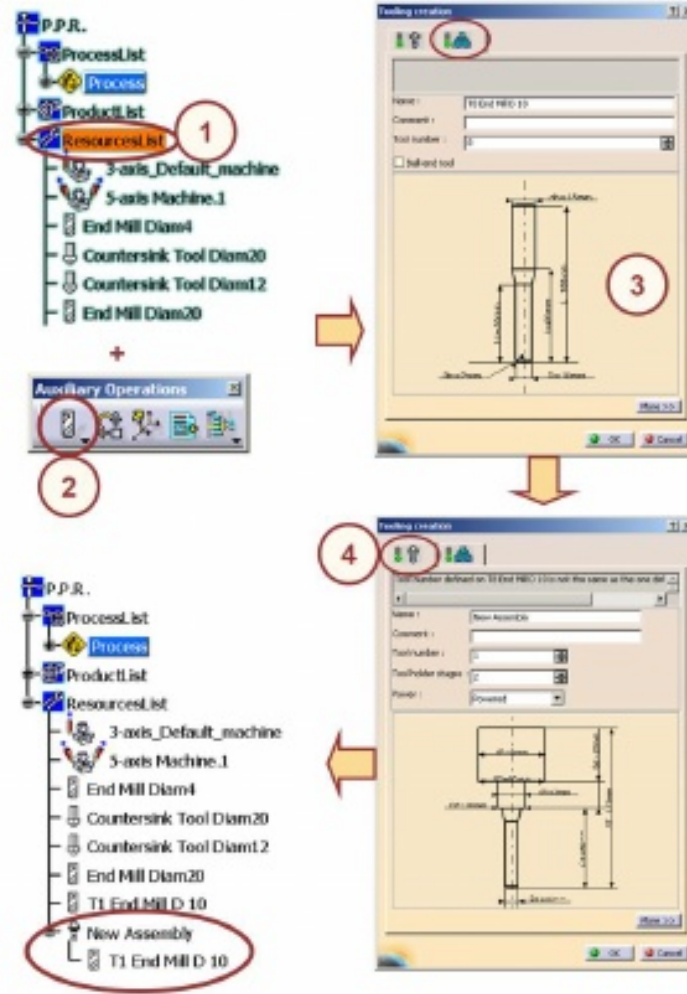
If the name of the assembly is not typed in the Name field, then only the tool will be created.

Feedrates Computation

You will learn what is Feed& Speed and how to compute Feedrates.

- **Feedrates: Introduction**
- **How to compute Feedrate**

Depth of cut in inches

Spindle speed in rpm or cutting speed in surface feet per minute (sfpm)

Feed in inches per spindle revolution
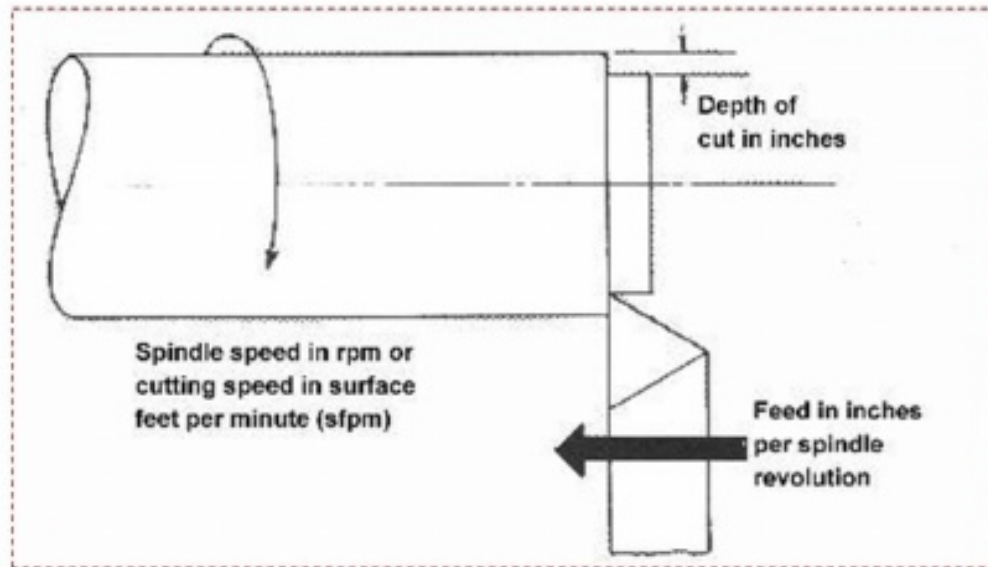
Feedrates: Introduction

Feedrate is the distance traveled by the cutting tool or workpiece in unit time and Speed is number of revolutions of the cutting tool or workpiece per unit time.

Cutting conditions (feed/tooth and cutting speed) can be included in a tools catalog. This data is converted into machining feedrate and spindle speed parameters to be used in machining operations by means of formula.

In the Feeds and Speeds tab page of milling operations, the Rough or Finish quality of the operation and the tool data are taken into account for computing the feeds and speeds.

When a tool is selected for an operation, spindle speed (N) and machining feedrate (Vf) are computed using the following formula:

N (in rev/mn) = Vc / (D * PI)

where:

D = tool diameter for milling/drilling in mm

Vc = cutting speed of the tool or insert.

For turning operations, N is automatically set in mm/min with the value of the insert's cutting speed.

Vf (in mm/rev) = Sz * N * Z

where:

Sz = feedrate/tooth on the tool

N = spindle speed in rev/min

Z = number of teeth on the tool (MFG_NB_OF_FLUTES) or 1 for a lathe insert.

How to Compute Feedrate

A. **Access to feedrate tab page**

B. **Feedrate Definition:**
- ◆ Activate « Automatic compute » if you want to compute feedrate from tool values
- ◆ Deactivate « Automatic compute » if you want to type your own values
- ◆ Activate « Transition » you can locally set the feedrate for a transition path to a machining operation from other machining operation or from a tool change activity.
- ◆ Select the feedrate unit: linear or angular

C. **Definition of Feedrate reductions in corners:**
- ◆ Reduction rate: feedrate in the corner = X % of machining feedrate
- ◆ Minimum angle: the feedrate will be reduce only in corner with an arc angle bigger than this value
- ◆ Maximum radius: no reduction of feedrate for corner with radius bigger than value
- ◆ Distance before / after corner: where start/stop the reduction feedrate

D. **Spindle speed:**
- ◆ Activate « Automatic compute » if you want to compute feedrate from tool values
- ◆ Deactivate « automatic compute » if you want to type your own values
- ◆ Activate Spindle output to Key the name of the assembly
- ◆ Select the feedrate unit: linear or angular

E. **Define quality:**
- ◆ Select Rough or Finish to load on the tool the appropriate set of feeds & speeds
- ◆ Click compute to update the values in the operation feedrate tab page
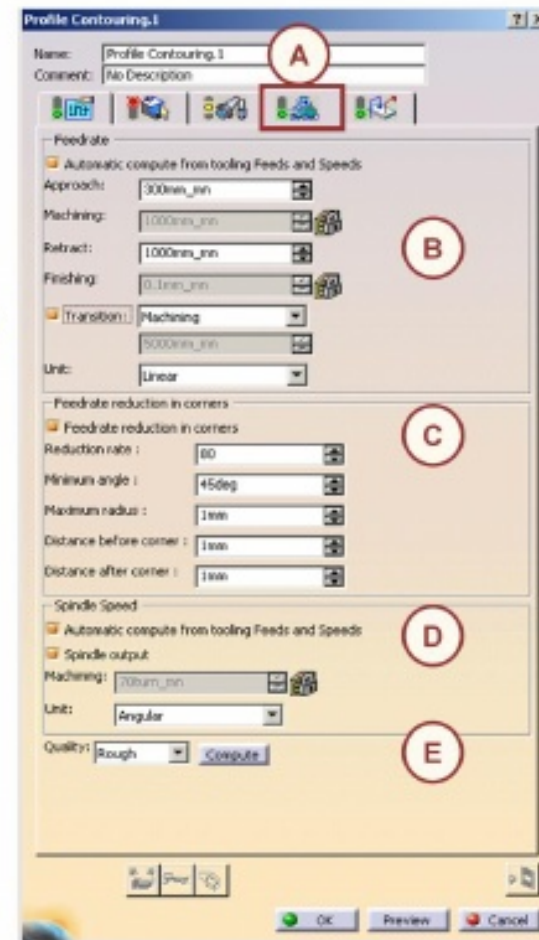
Macro Motions

Macro Motions are the tool motions outside the material.

- **Introduction**
- **Definition**
- **Catalogs for Macro Management**
- **How to Store Macros in Catalogs**

Macro Management

Macro	Name	Mode
Approach	Approach.1	None
Retract	Retract.1	Build by user
Clearance	Clearance.1	To safety plane
Linking Retract	Linking.1	Build by user
Linking Approach	Linking.1	Build by user
Return in a Level Retract	Return in a Level.1	Build by user
Return in a Level Approach	Return in a Level.1	Build by user
Return finish pass Retract	Return Finish Path.1	Build by user

Current Macro Toolbox

Definition | Options

Mode: Circular horizontal axial
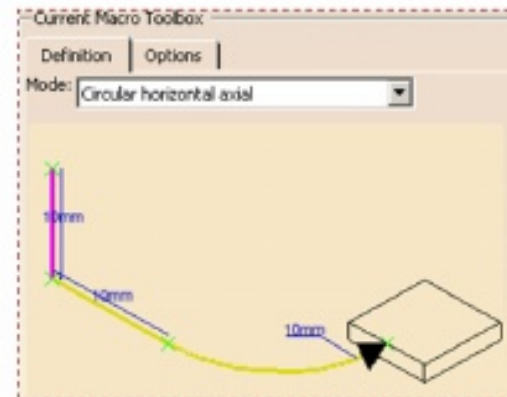
Macro Motion: Introduction

The NC Macro option provides features that enhance productivity.
The non-working motions are controlled by macros. Thus the tool idle time
in machining is reduced.
Tool damages either by collision or plunging are avoided using macros.
Different types of macros are used according to the machining processes.
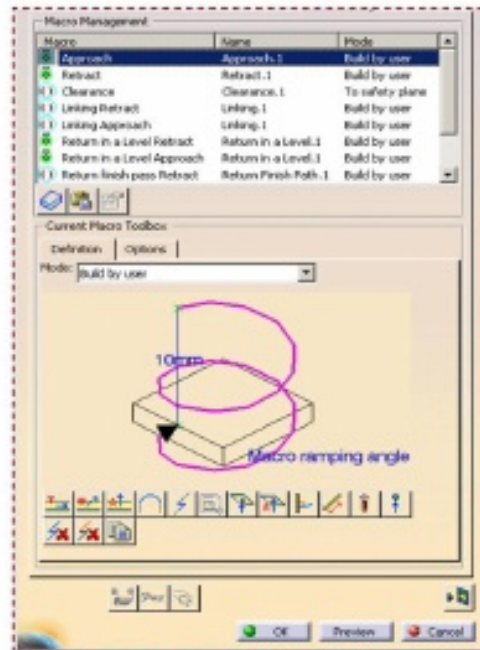
You can use the pre-defined macros or you can create your own macro as per the requirement.

Macro Motion Definition (1/2)

Macro definition or modification:

(A) **Click on an element to affect geometry**

(B) **At each intersection you can add PP word instructions (green cross)**

(C) **MB3 on a motion:**
- ◆ **Deactivate it**
- ◆ **Define Feedrate**
- ◆ **Delete it**
- ◆ **Insert a new motion after it**

The color of the line is according to the Feedrate:

Yellow : Approach
White : Local, Finishing
Green : Machining
Blue : Retract
Red : Rapid
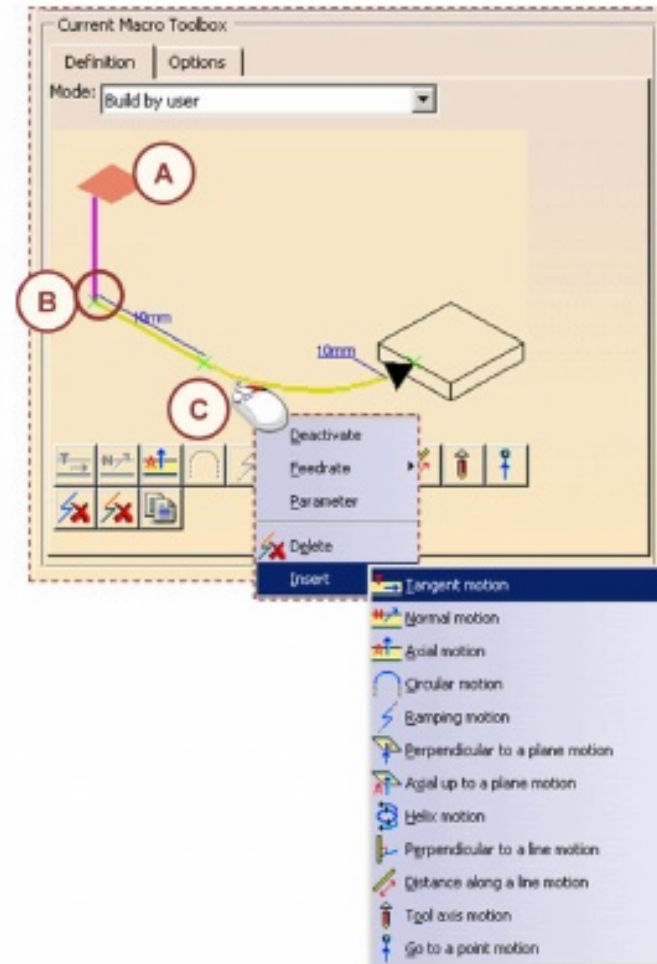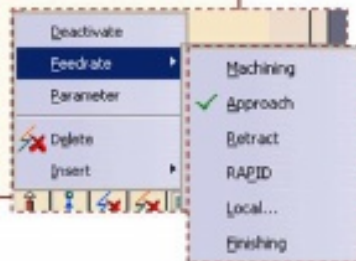
Macro Motion Definition (2/2)

Macro Build by user:

Icon	Description		Icon	Description
	Tangent			**Axial up to a plane**
	Normal			**Normal to line**
	Axial			**Along a line**
	Circular			**Along tool axis**
	Ramping			**To a point**
	Helix			**Erase All**
	Add PP Word instruction			**Erase Selected motion**
	Up to a plane and normal to it			**Copy Approach or Retract macro on all approach or retract motions of the other macros**

Catalogs for Macro Management

The **Catalog** is the way to store also the standard NC Macros. These catalogs are defined directly from Macro definition tab page in CATIA V5.
The stored macros are accessible directly from the same dialog box during Machining operation definition.
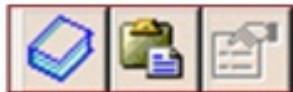
About a Macro Catalog:

- A Setup Catalog of macro is including in CATIA installation under \intel_a\Startup\Manufacturing\Macros directory

- During a CATIA V5 session, you can access several Macros Catalog during operations creation in a single Part Operation

- Macro Catalogs are CARTIA V5 standard catalog, so you can edit and organize them as you want.

How to Store Macros in Catalogs

1. **Create a Machining Operation**
2. **Define your macro (Parameters, Name, Comment)**
3. **Store it in a catalog**
   - **Select Create a new catalog or**
   - **select "…" button to Update an existing Catalog Or**

2bis - Retrieve a Macro from a catalog

   Select your Macro Catalog, type of macro, Macro

Axial Operations

You will learn how to create Axial Operations.

- Creating an Axial operation: General Process
- Creating an Axial operation
- Strategy
- Geometry
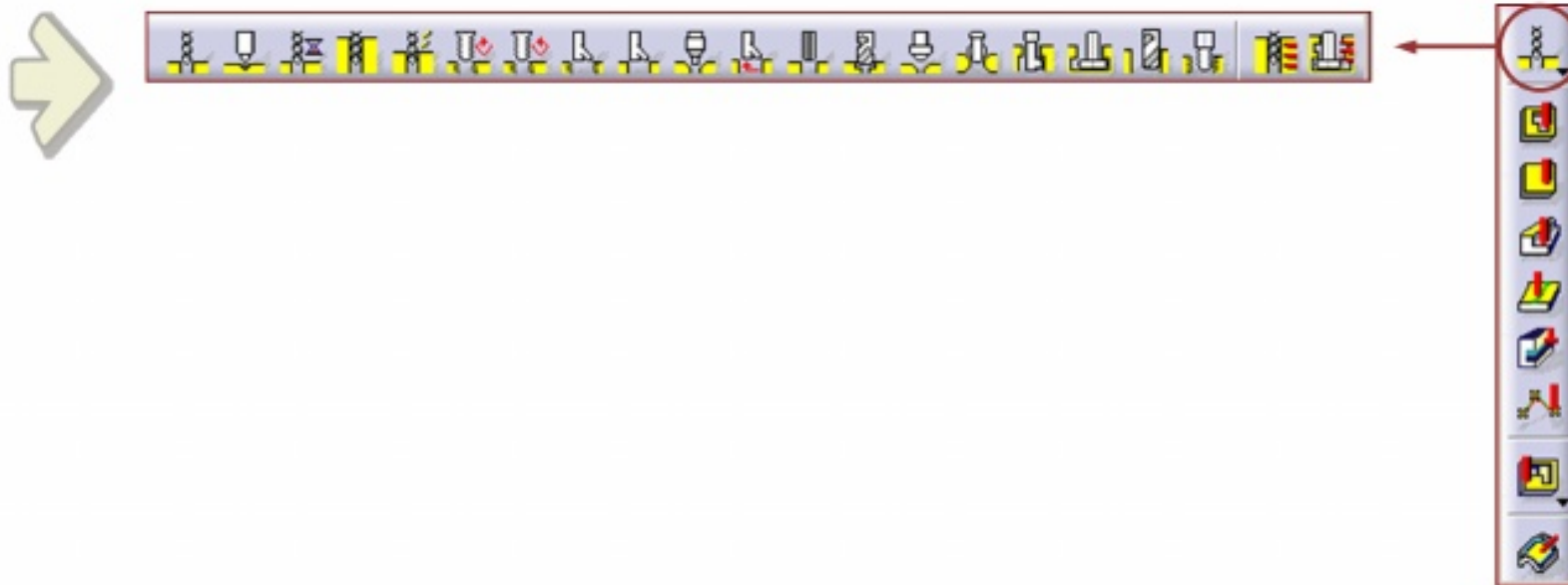
Various Axial Operations

You can create Axial machining operations on a single point or on a pattern of point.

Following are the Axial Machining Operations:
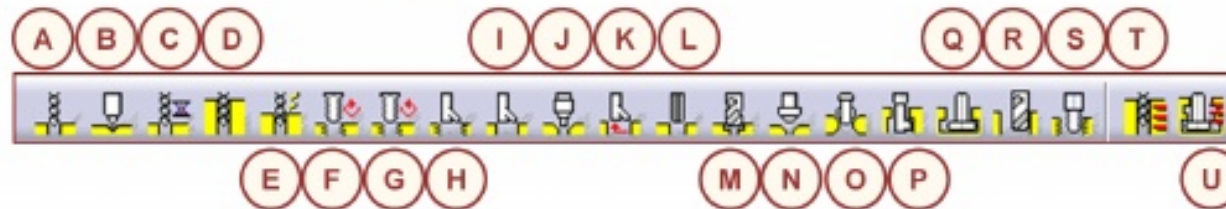
A. Drilling	K. Boring Spindle Stop
B. Spot Drilling	L. Reaming
C. Drilling Dwell Delay	M. Counter Boring
D. Drilling Deep Hole	N. Counter Sinking
E. Drilling Break Chips	O. Chamfering 2 Sides
F. Tapping	P. Back Boring
G. Reverse Threading	Q. T Slotting
H. Thread Without Tap Head	R. Circular Milling
I. Boring	S. Thread Milling
J. Boring and Chamfering	T. Sequential Axial
	U. Sequential Groove

Creating an Axial Operation: General Process

You will see one axial operation in detail.

1 Click **Axial Operation** Icon

2 Select the operation to be created

3 The new Part Operation is created after the current one. The Operation dialog box displays to edit it.

4 Define the Operation geometry and parameters in the dialog box.

5 Click OK to create the operation

Multi-edition possible of a parameter on a set Machining Operation of a same type

P.P.R.
ProcessList
Process
Part Operation.1
Manufacturing Program.1
Tool Change.1 Drill_Default_Tool
Drilling.1

Creating an Axial Operation

A — Name of the Operation + Comments

This comment will be generated in the APT Source with the **PPRINT** prefix at the beginning of the operation

B — Edit Cycle allows to define your own drilling cycle for APT generation

You can edit it or modify it
- Using PP word Assistant

C — Replay Preview Button
- Allow to check consistency between the geometry to machine, the tool and parameters of the operation
- Information message is displayed
  - Check Tool diameter / Geometry diameter to machine
  - Tool pitch and tool way of rotation / Threaded Geometry

Manufacturing Information: Tool diameter greater than the diameter to machine — OK

D — Replay and / or Simulate the operation tool path (See dedicated Job Aid 'Replay a Tool Path')

Drilling.2 — Name: Drilling.2 — Comment: No Description

Power: From Tool Assembly
Approach clearance (A): 1mm
Depth mode: By tip (Dt)
Breakthrough (B):
Plunge mode: None
First compensation: 1
Compensation application mode: Output point
Automatic ROTABL:
Output CYCLE syntax:

OK — Preview — Cancel
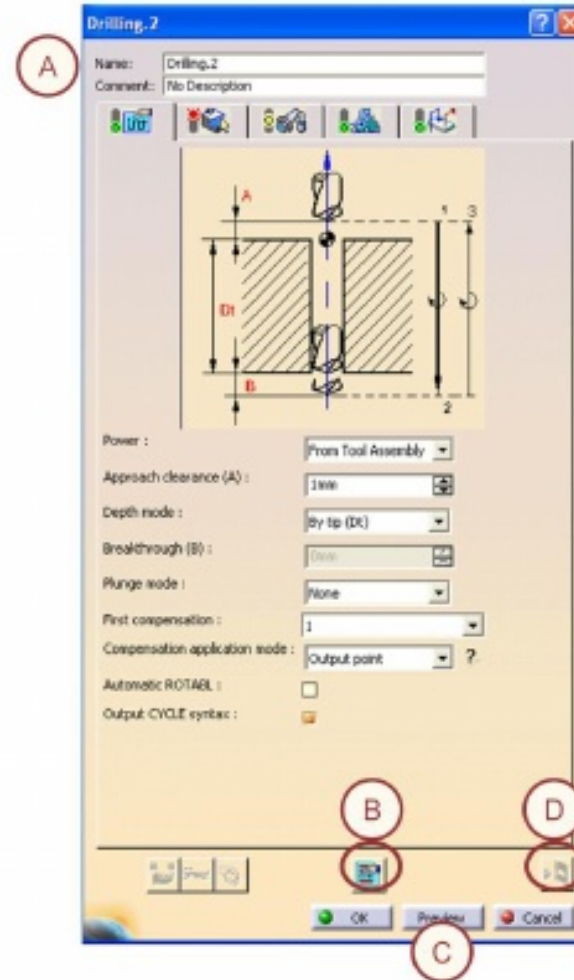
Creating an Axial Operation: Strategy

Strategy tab page detailing:

A. The icon describes the tool path and the parameters impacting the cycle.

It is updated when you change a parameter modifying the tool path

Depth defined by Shoulder or by Tip
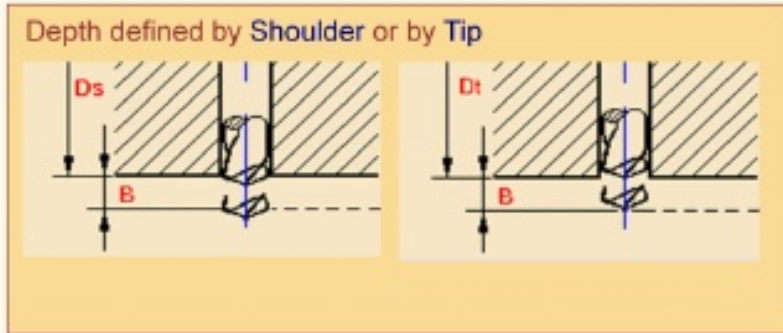
B. Define parameters for the operation

- ◆ Power: From Tool Assembly / Fixed / Powered
- ◆ Approach Clearance offset (A)
- ◆ Depth Computation mode
- ◆ Plunge Options (See dedicated Job Aid 'Plunge Options')
- ◆ Breakthrough value for Through Hole (B)
- ◆ Tool Compensation number
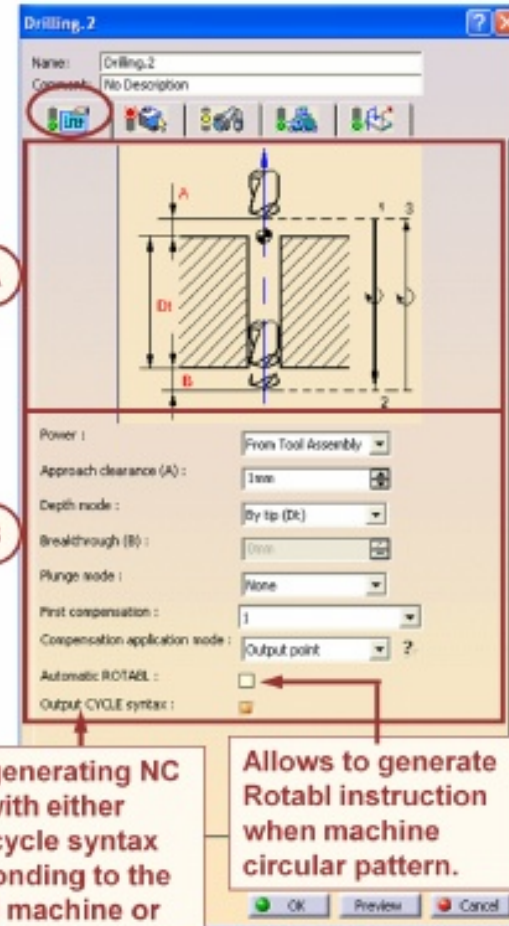
Allows generating NC output with either drilling cycle syntax corresponding to the selected machine or GOTO points

Allows to generate Rotabl instruction when machine circular pattern.

Pattern Management- Hole Selection

You will learn,how to create a Machining Pattern.

- Creating a Machining pattern: General Process
- How to Create a Machining Pattern
- Power Search
- How to Search Axial Features

Creating a Machining Pattern: General Process

It allows to create a Machining Pattern in the machining operation definition or by clicking on the machining pattern icon

1 Select the red hole depth representation in the sensitive icon or click icon.

2
a. Select a Design Pattern or an already created Machining Pattern in the Pattern Selection dialog box or
b. Select directly the geometry in the CATIA screen to add it to the Machining Pattern
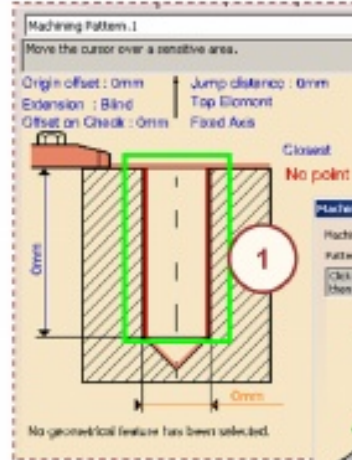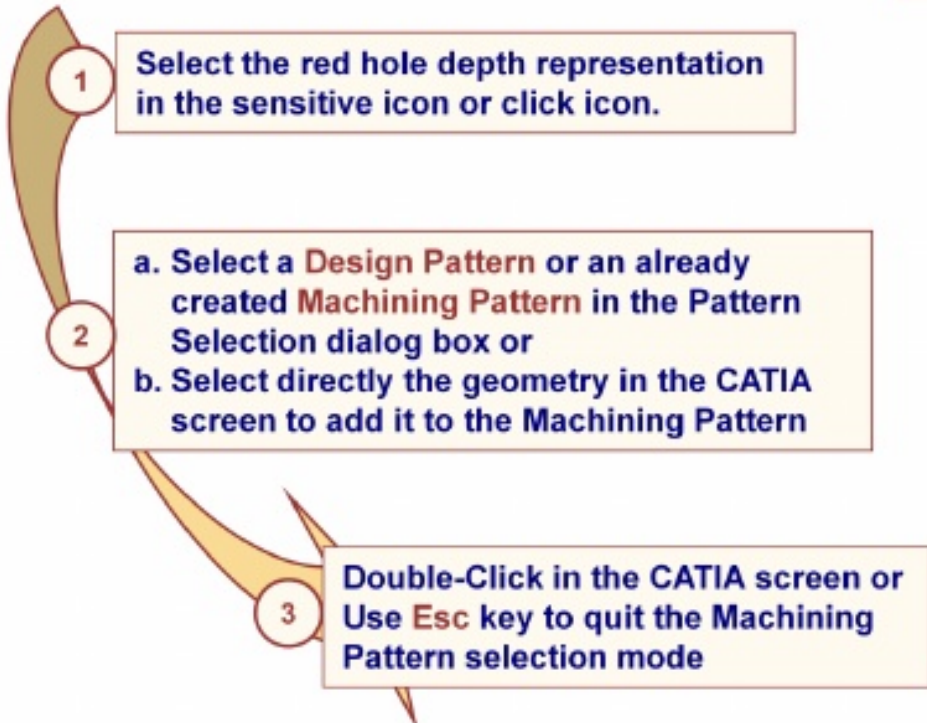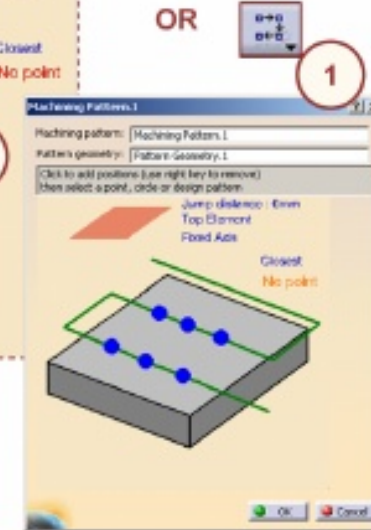
3 Double-Click in the CATIA screen or Use Esc key to quit the Machining Pattern selection mode

OR

A Machining Pattern can include Design Patterns and/or individual holes and/or other Machining Patterns and/or point and/or circular edges

2a

Pattern Selection
- Machining/Design Patterns
  - RectPattern.1
  - RectPattern.2
- Pattern Geometry.1
  - Machining Pattern.1
    - Drilling.1
      - T1 Drill D 10

2b
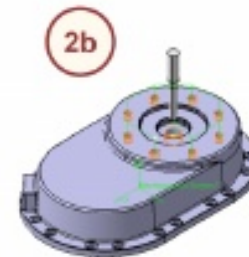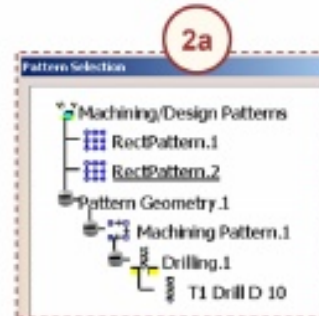
How to Create a Machining Pattern (1/2)

You can predefine your Machining Pattern and can be reused in different operations.

(A) The **Jump Distance** is used to specify an offset on the top of the holes that will be applied for the transition paths between two holes.
This transition path is perform in **RAPID** mode

(B) The 3 ways for ordering pattern points (MB3) are **Manual, Closest or By Band**

Manual: Successively select the points in the order you want them sequenced

Closest: The pattern point closest the first point is given the next sequence number,
the next closest to that is given the next sequence number and so on

By Band: Define the mode Zig zag or One way & the width of the bands
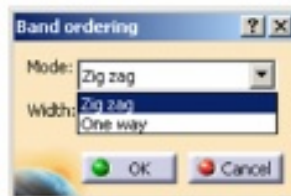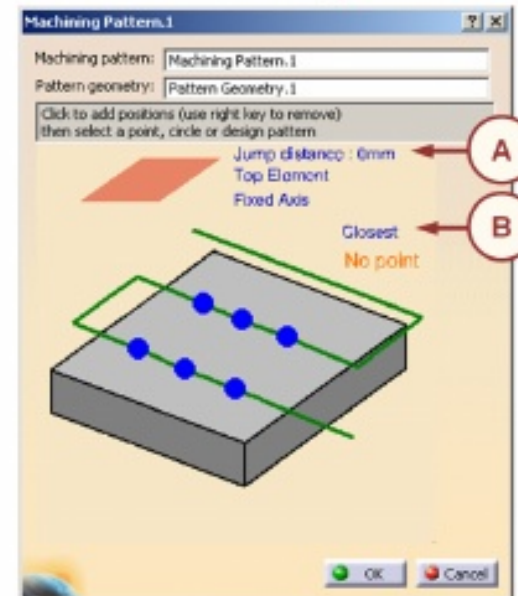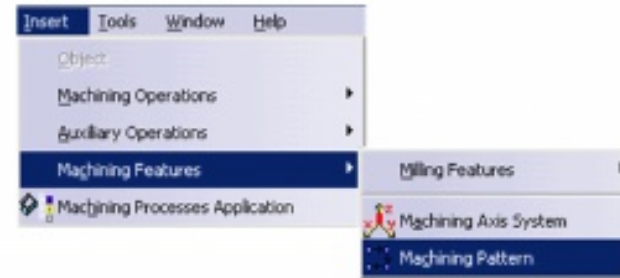
How to Create a Machining Pattern (2/2)

(C) **Hole Selection**
- ◆ **Select Hole and pattern one by one or**
- ◆ **MB3 on "No Point" and select the option**
- ◆ **Remove all selected holes/patterns**
- ◆ **Remove one position**
- ◆ **Deactivate position from X to Y**
- ◆ **Find Features through Faces: you select all the circles on faces**
- ◆ **Reverse Ordering**
- ◆ **Analyze: to visualize the different entities of the machining pattern**
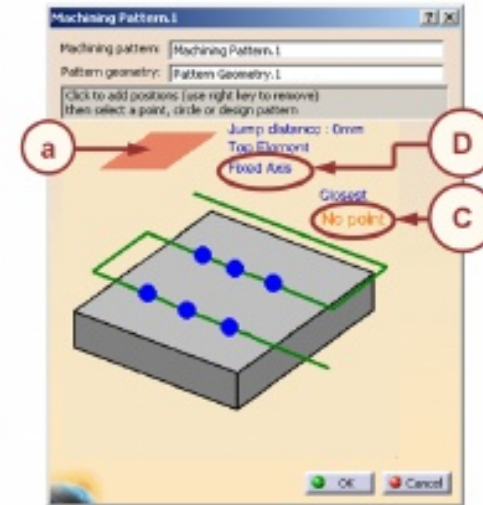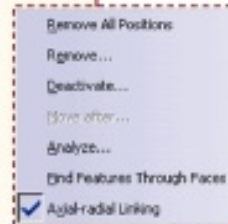
(D) **Three Ways to define Tool Axis (MB3)**
- ◆ **Fixed, Variable or Normal to Part Surface**
  - • **If you select Normal to PS, you must define the Part Surface by selecting (a)**
  - • **If you select Variable, using contextual menu on the part (mb3 on arrow) (b) you can define a axis direction for each point**
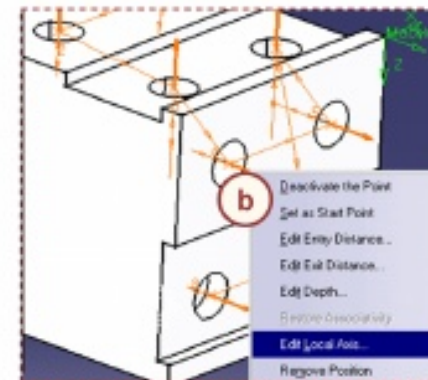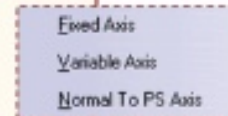
Local modification of a Pattern of Points
A contextual menu is available at each point of the Machining Pattern using MB3. The following actions are available as shown.

Power Search: General Process
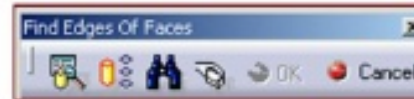
Power Search is the selection of Hole Design Feature.
It allows to search hole on the part.

Select « find features through faces » in contextual menu

1. Select range of diameters
2. Select the faces
   Select a reference feature on the part if you want to apply all it
   properties to all positions (optional)
3. Make queries on features (optional)
4. Click search icon
5. Click OK
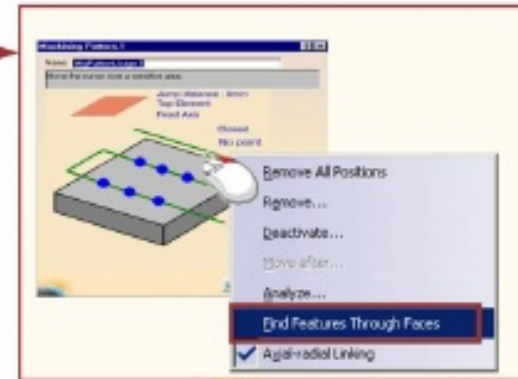
Supports Mirror and Transformation operators
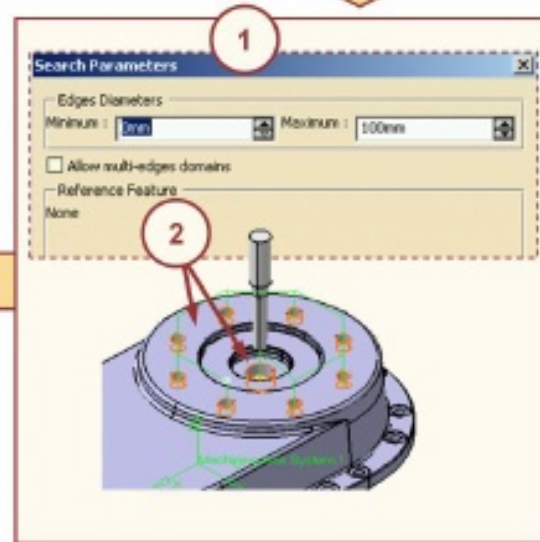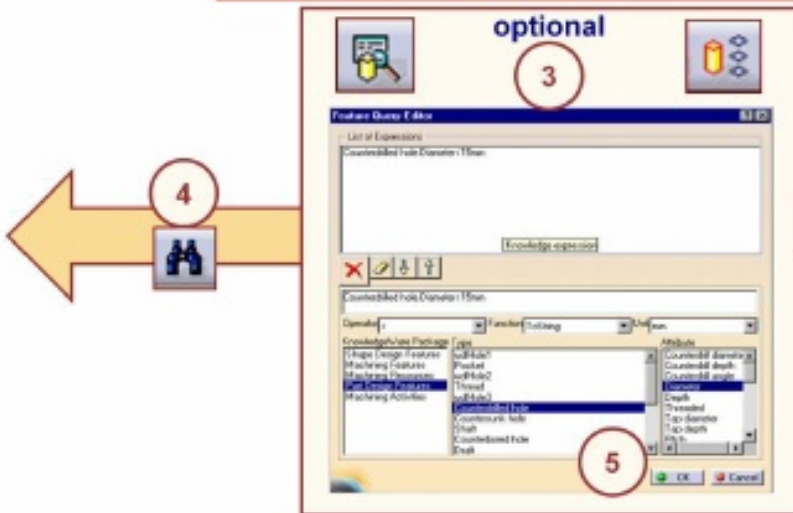
optional

How to Search Axial Features
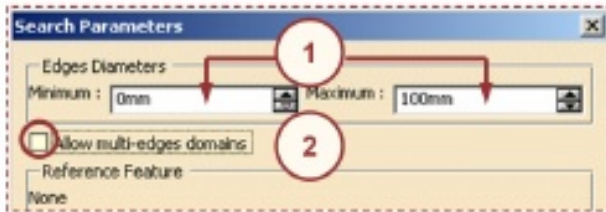
1. Type minimum diameter and maximum diameter to search a specific diameter, type the same value

2. Activate the option if the hole edge is composed of several elements

1. Click the icon and select a feature on the part or in the specification tree
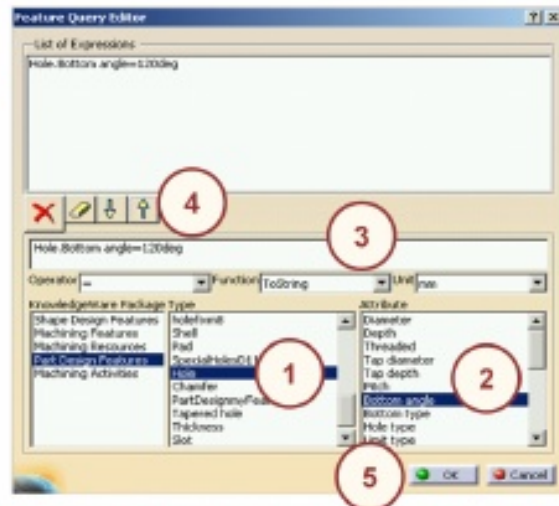
1. Select the feature (Hole)

2. Select the attribute

3. Complete the formula for the query

4. Add/remove it to the list

5. Confirm
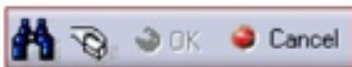
Click search and OK to validate remove all the positions or cancel

Process Views

You will learn the different Process Views.

- **Introduction to Process Views**
- **Manufacturing View details**
- **Process Table details**

Introduction to Process Views

You have different ways to visualize the Machining entities in the CATProcess:

First based on the **PPR**: ProcessList which is a sequential view on the Machining Program display as a tree structure
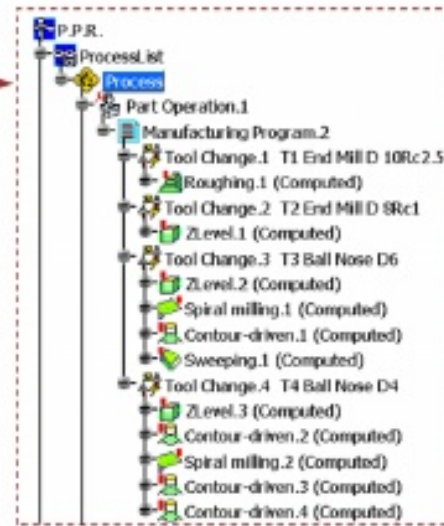
The second one based on the Manufacturing view: which allows you to visualize your Machining Program sort by Machining object and see the element attached to this object (Pattern, Tools, Operation type, Features)
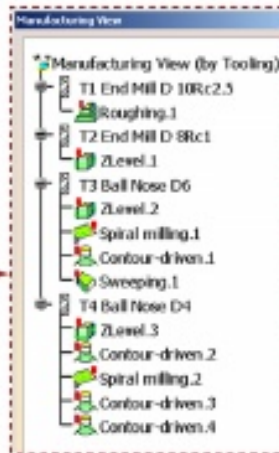
The Last one, the Process Table: which is a tabular view of the Process, or a given Part Operation or a given Manufacturing Program.
It provides an alternative view to the PPR tree.

Manufacturing View Details (1/3)

Each time you create an operation, the system creates a new machining feature which is the geometry (machining pattern, surface) machined in this operation.

This machining feature is available for any further operation.
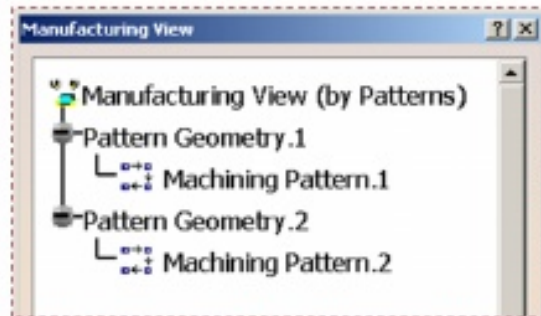
The Machining/ Machinable Features are directly accessible in the combo list in the definition operation dialog box.

85
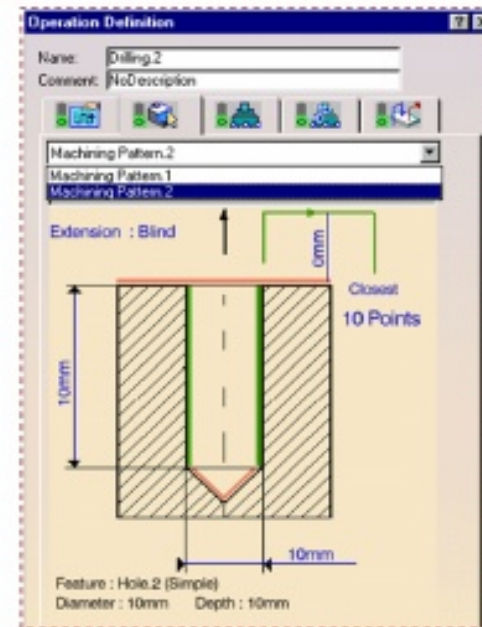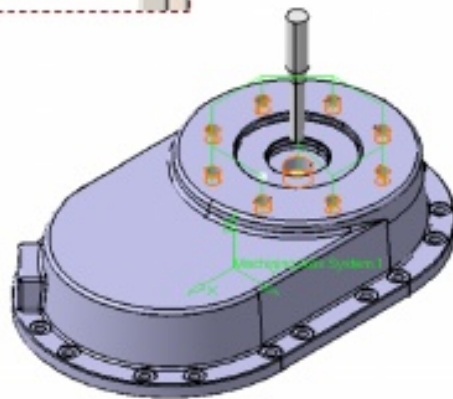
Manufacturing View Details (2/3)

Using Manufacturing View, you can visualize:

A. **Features: basic geometry of the design and Relation (Check and Rules)**

B. **Patterns: design and machining patterns in PO**

C. **Machining operations in PO with associated tools**

D. **Machining operations in PO sorted by tool**

E. **Machinable Features: Predefined set of geometry**
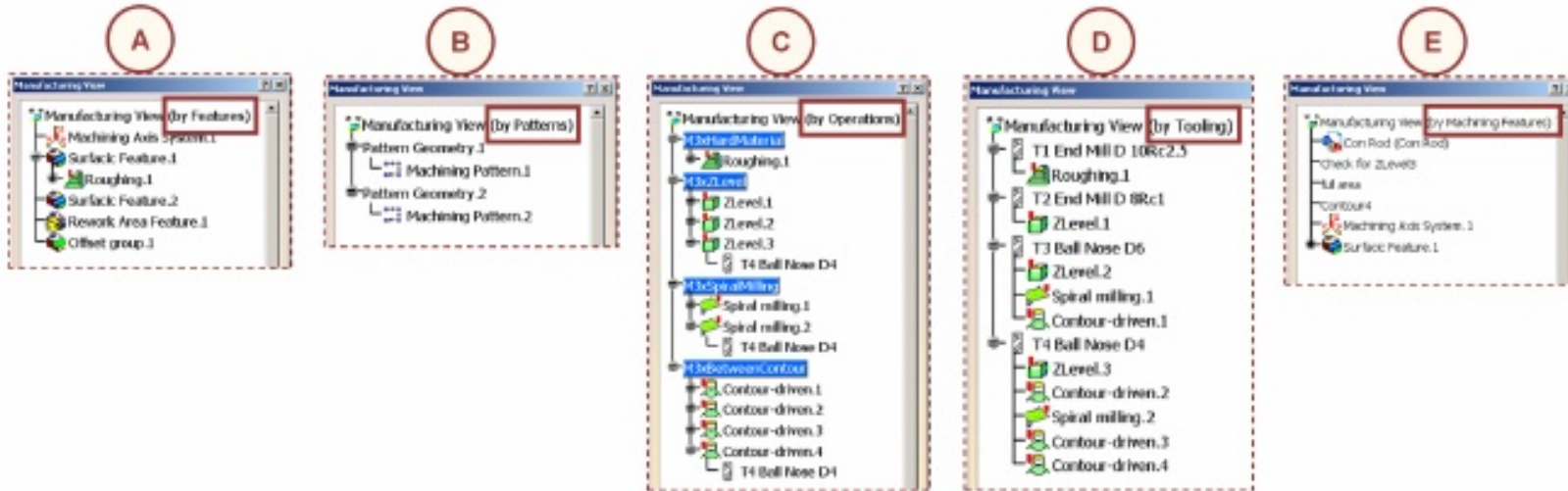
You can use 'Advanced Sort and Filtering' option if your view is sorted by Machining Features.

Manufacturing View Details (3/3)

Advanced Sort and Filtering:
This functionality allows you to sort Machinable Axial Features and Machinable Milling Features.

Machinable Axial Features are axial features created by Prismatic Machining Preparation Assistant and Machinable Milling Features consist Prismatic Machining Area and Prismatic Rework Area features.
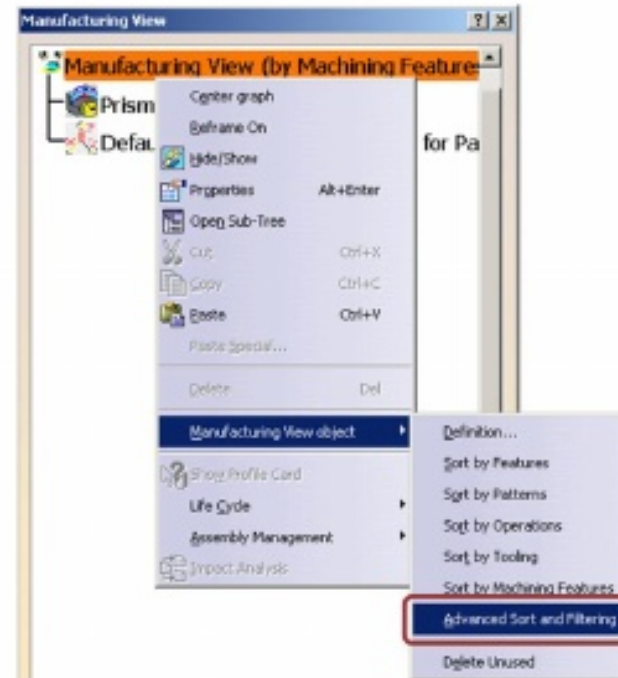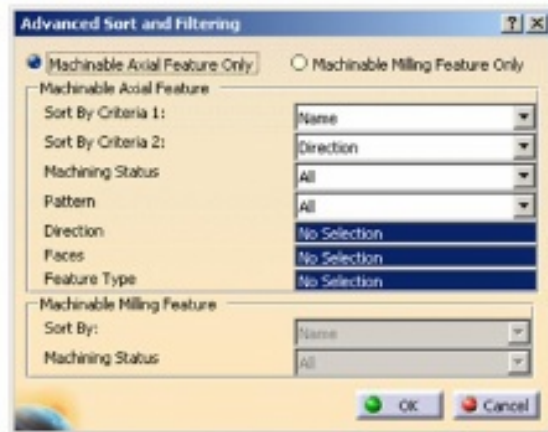
You can sort either Axial features or Milling Features.

- **For Axial features, you can sort by different criteria and filter by Machining Status, Pattern, Direction, Faces or Feature Type**
- **For Milling features, you can sort by name and filter by Machining Status.**

Unused Machinable Features can be deleted using 'Delete Unused' in contextual menu.

Process Table Details (1/2)

Introduction:

This window gives you a table view of your Part Operations, Manufacturing Program and Machining operations with associated parameters.

Like for the Manufacturing view,

- **Selecting an entity in the view will highlight the corresponding operation in the other views (PPR tree and the Manufacturing view).**
- **You can edit an operation directly from this table and have the same contextual menu as in the PPR tree.**

You can access to this window using the dedicated icon or Contextual menu on the Part Operation or Manufacturing Program

Access to predefined table views (columns order and filter)

Number	Name	Type	Machining...	Spindle...	S	Resources name	C	Status	L...	Computed	Packed	W...	Specified tin
	Part Operation.1	Part Operation	-	-	-	3-axis_Machine_...	-	-	-	-	-	No	5s
	Manufacturing Program.2	Manufacturing Program	-	-	-	-	-	-	-	-	-	No	5s
1	Tool Change.1	Tool Change	-	-	-	-/T1 End Mill D 10...	-	-	-	-	-	No	5s
2	Roughing.1	Roughing	1400mm_mn	4000tu...	-	-/T1 End Mill D 10...	-	Not ...	No	Computed	No	No	5s
3	Tool Change.2	Tool Change	-	-	-	-/T2 End Mill D 8Rc1	-	-	-	-	-	No	5s
4	ZLevel.1	ZLevel	1600mm_mn	12000t...	-	-/T2 End Mill D 8Rc1	-	Up T...	No	Computed	No	No	5s
5	Tool Change.3	Tool Change	-	-	-	-/T3 Ball Nose D6	-	-	-	-	-	No	5s
6	ZLevel.2	ZLevel	2000mm_mn	12000t...	-	-/T3 Ball Nose D6	-	Up T...	No	Computed	No	No	5s
7	Spiral milling.1	Spiral milling	2000mm_mn	12000t...	-	-/T3 Ball Nose D6	-	Up T...	No	Computed	No	No	5s
8	Contour-driven.1	Contour-driven	4000mm_mn	20000t...	-	-/T3 Ball Nose D6	-	Up T...	No	Computed	No	No	5s
9	Tool Change.4	Tool Change	-	-	-	-/T4 Ball Nose D4	-	-	-	-	-	No	5s
10	ZLevel.3	ZLevel	3000mm_mn	24000t...	-	-/T4 Ball Nose D4	-	Up T...	No	Computed	No	No	5s
11	Contour-driven.2	Contour-driven	2000mm_mn	24000t...	-	-/T4 Ball Nose D4	-	Up T...	No	Computed	No	No	5s
12	Spiral milling.2	Spiral milling	3000mm_mn	24000t...	-	-/T4 Ball Nose D4	-	Not ...	No	Computed	No	No	5s
13	Contour-driven.3	Contour-driven	3000mm_mn	24000t...	-	-/T4 Ball Nose D4	-	Up T...	No	Computed	No	No	5s
14	Contour-driven.4	Contour-driven	3000mm_mn	24000t...	-	-/T4 Ball Nose D4	-	Up T...	No	Computed	No	No	5s

Process Table Details (2/2)

How to use the Process Table:

Right-click in the Process Table to access a number of commands
- ◆ **Column Filter**
- ◆ **Column Order**
- ◆ **Select by String**

These commands allow you to customize the table to your needs.

📑 **Column Filter:**
Select the columns that you want to include in the Process Table
You can use the Ctrl and Shift keys to make multiple selections.

📑 **Column Order:**
You can change the order of the filtered columns in the Process Table by selecting a line and moving it in the list by means of the Up / Down buttons.

📑 **Select By string:**
You can use the pop-up that appears to search for any character string in the Process Table
(for example: tool change).
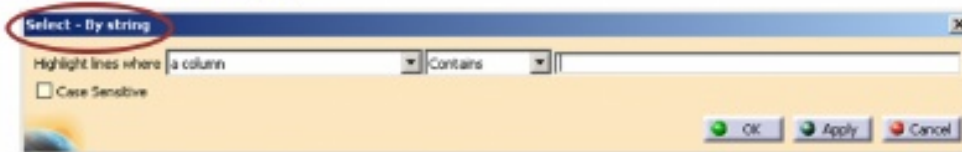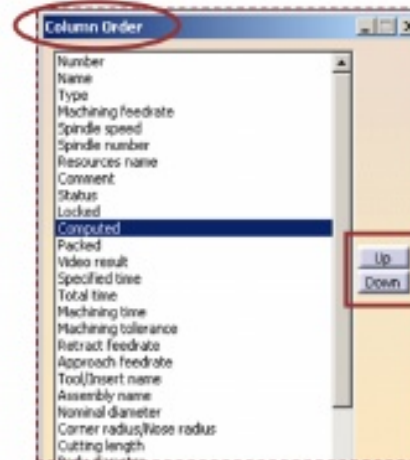
Geometry Wizard (Edge, Face)

You will learn how to select Edge and/or Face which is necessary for machining operations.

- **Introduction to Geometry Wizard**
- **Edge Selection**
- **Face Selection**
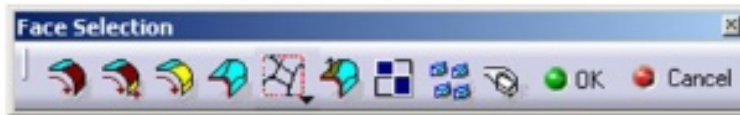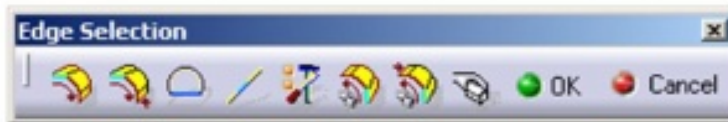
Introduction to Geometry Wizard

- The Edge Selection toolbar contains commands to help you to select edges of contours when specifying geometry in machining operations.

- The Face Selection toolbar and Tools Palette appear when face selection is necessary for machining operations.
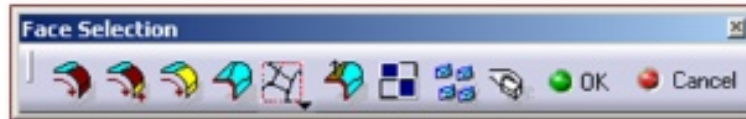
- In some cases when automatic propagation is interrupted, a label appears at the extremity of the last selected edge. For example:
  - **Next:** This means that the maximum number of steps forward has been reached.
  - **Angle:** This means that the maximum angle is not respected or there is an ambiguity.
  - **Tolerance:** This means that the maximum gap is not respected.
  - **Closed Loop:** This means that the contour is closed

How to use Edge Selection (1/2)

1. Select the mode of navigation:

- ◆ **Link type (way of link elements between two not consecutive edge selection)**
- ◆ **Number of elements for automatic navigation**

2. Propagation Domains:

By default, only the edges included in the current Body (or Geometrical Set) can be selected. You can add other bodies by clicking the Add button and selecting new bodies in the 3D viewer. You can remove selected bodies by right-clicking the Propagation Domains area and selecting the Reset contextual command.

3. During automatic propagation, if there are more that one possible edges for selection, the best candidate is selected according to the following criteria:

- ◆ **The gap between the last selected edge and the candidate edge must be less than the Maximum gap.**
- ◆ **The angle between the tangent of the candidate edge and the tangent to the last selected edges must be less than the Maximum angle. If there still more that one candidates, the one that makes the smallest angle is preferred.**

Options

Link types : Automatic   ①

Reverse Propagation

Propagation Domains

Add   ②

Maximum gap | 0.1mm

Maximum angle | 45deg   ③

Max Steps Forward | 10

Steps Back | 1

Close
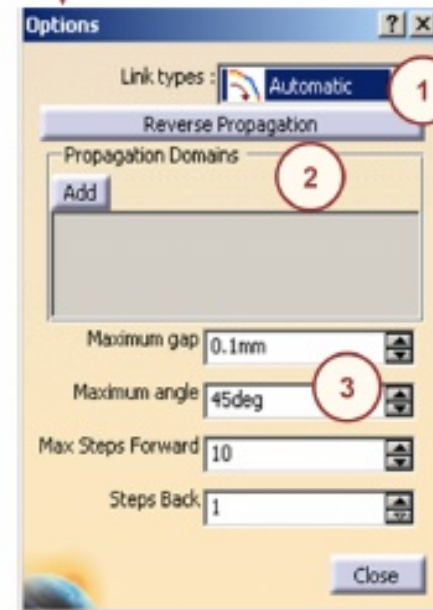
How to use Edge Selection (2/2)

◆ **Automatic edge selection**
- **Action: Select one element then click the icon**
- **Result: Select the X next edges in the indicated direction**

◆ **Automatic edge selection until selected element**
- **Action: Select the first edge, click the icon and select the last edge**
- **Result: Select the X edges between the two selected edges**

◆ **Insert Line**
- **Action: Click the icon and select the two points**
- **Result: a line is inserted between the two points**

◆ **Close Contour**
- **Action: Click the icon**
- **Result: a line is inserted between the two extremity of the contour**

◆ **Delete edges**
- **Action: Click icon**
- **Result: Delete the Y last edges**

◆ **Delete edges until element**
- **Action: Click icon and select the last edge you want to keep**
- **Result: Delete the last edges until the selected edge**

◆ **Delete contours**

◆ **Validate selection or cancel**   ● OK   ● Cancel

Options

Link types : Automatic

Reverse Propagation

Propagation Domains
Add

Maximum gap 0.1mm

Maximum angle 45deg

Max Steps Forward 10   **X**

Steps Back 1   **Y**

Close
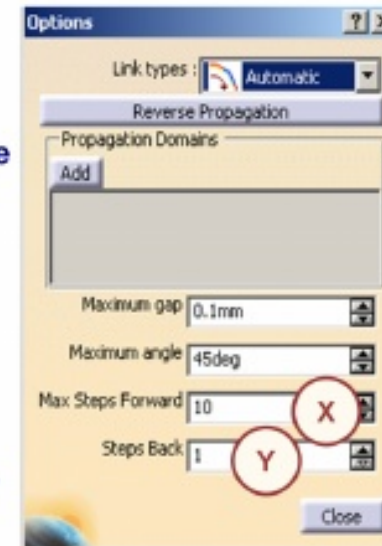
How to use Face Selection (1/2)

The Face Selection toolbar contains commands to help you to select faces
 when specifying geometry in machining operations.

♦ **Navigate on Belt of Faces** icon allows you to select all faces that are adjacent to the one you have selected.

   ▪ Select two adjacent faces and click the icon. All adjacent face are selected.

♦ **Navigate on Faces** Until a Face icon allows you to select all faces that are adjacent between start faces and a stop face.

   ▪ Select two faces that are adjacent (to give the direction of selection) and then click the icon.

   ▪ select a third face where you want selection to end.

♦ **Navigate on Faces** icon allows you to select all faces which are tangent to a selected face.

   ▪ Select a face and then click this icon.

♦ **Preview the Contour** icon allows you to highlight the contour of selected faces.

♦ **Select Faces in a Polygon Trap** icon allows you to select all faces that are situated entirely within a polygon.

   ▪ Select the icon.

   ▪ Click the places in the viewer where you want the corners of the polygon to be. Double-click to end corner definition.

♦ **Select Visible Faces in a Polygon Trap** icon allows you to select only the faces

   that are located entirely within a polygon and that are visible on the screen.

How to use Face Selection (2/2)

◆ **Select Normal Faces** icon lets you select faces that are:
  - **normal to a main axis.**
  - **parallel or perpendicular to a face that you select as reference.**

◆ **Retrieve Faces of Same Color** icon allows you to select all faces of a given color.
  - **Select a face of a given color and then click the icon. All faces of that color are selected.**
  - **Note that you can define the color of a face via the Edit/Properties menu item when the face is selected.**

◆ **Selection Sets** icon allows you to select faces belonging to previously created selection sets. This action is a shortcut to the Selection Sets item in the Edit menu.
  - **Click the icon and select the selection set you want to use in the displayed dialog box.**
  - **Press Close.**

◆ **Reset All Selections** icon.
  - **Click the icon to reset all selections made with the Face Selection toolbar.**

◆ **Accept / Cancel Geometry** Selections icon allows you to accept / refuse selected geometry and exit selection mode.

OK   Cancel
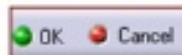
Tool Path Verification and Simulation

You will learn how to verify and simulate the Manufacturing Program.

- Accessing Replay and Simulation Tools
- Reply and Simulation Tools User Interface
- How to Replay a Tool Path
- How to Simulate Material Removal by Photo
- How to Simulate Material Removal by Video
- NC Manufacturing Verification NVG
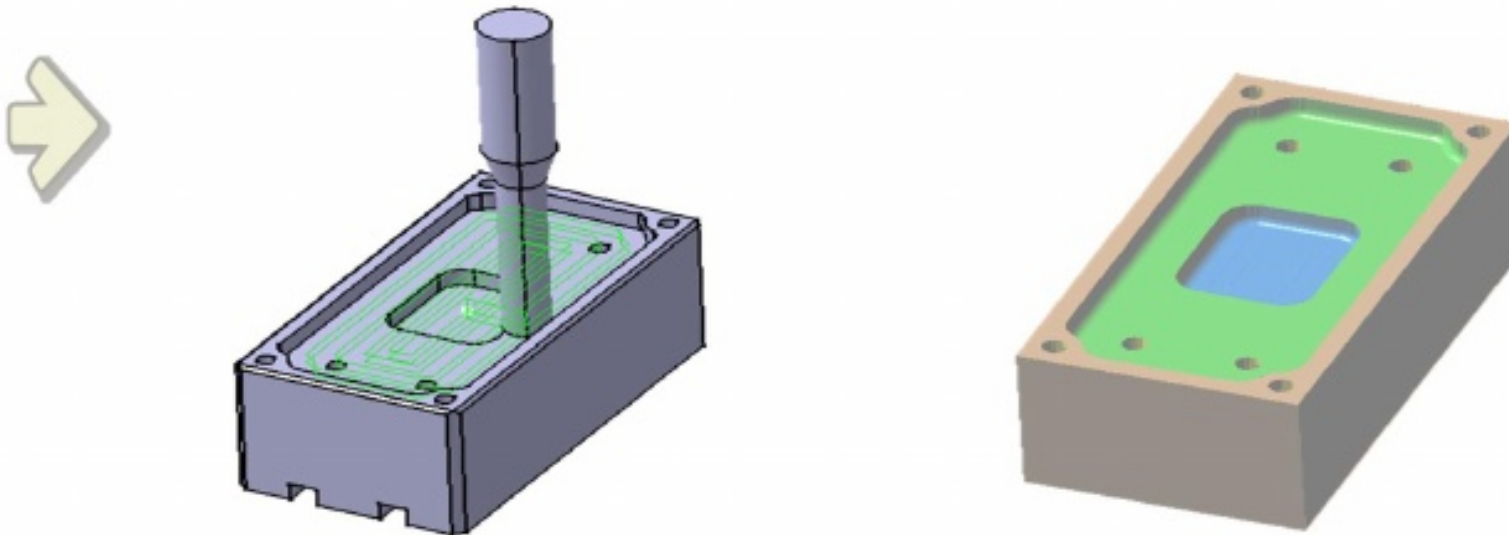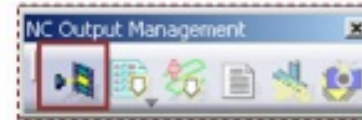
Accessing Replay and Simulation Tools

The **Replay** of the tool path displays tool trajectory and it can be run for:
- **A Manufacturing Program**
- **One or several Machining Operations**

A. **Select Manufacturing Program or Operation in the tree. Then Right-Click and select Replay Tool Path in the contextual menu** or

B. **Select the operation in the tree and click icon in the menu bar** or

C. **Edit the Operation and use Replay button**
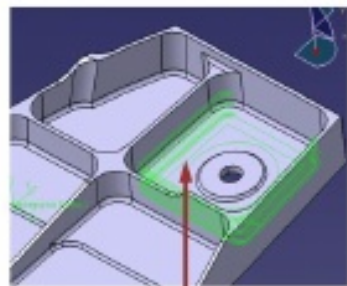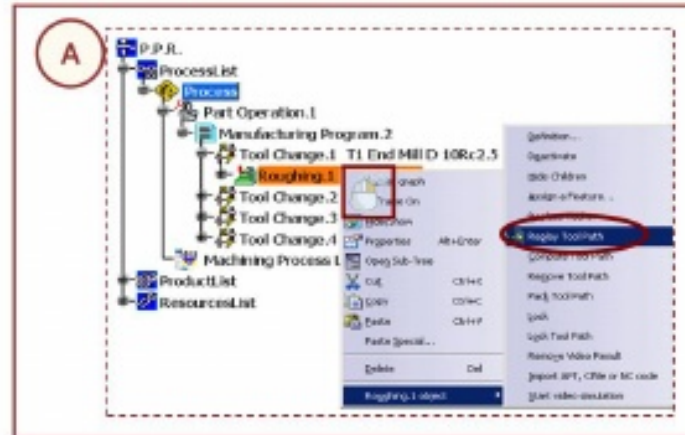
Tool path replay
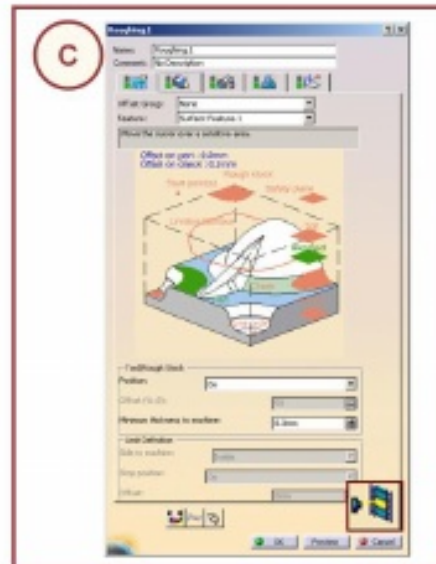
Replay and Simulation Tools User Interface (1/5)

Replay Dialog box:

From the Replay dialog box you can access to different functionalities:

Replay controls for:
- **Tool path replay**
- **Material removal simulation**

Tools for tool path replay

Tools for photo simulation

Tools for material removal simulation

Partial Tool Path Replay

Tools for machine tool simulation

Roughing.1

Tool animation

Replay positions
Start   1
Current 4599
End   4599

Feedrate =RAPID
X=-5.089mm Y=148.713mm Z=-0.818mm
I=0 J=0 K=1
Machining time = 15mn 2s   Total time = 17mn 19s

OK

Information on tool path:
- **point & vector coordinates**
- **Feedrate**
- **Machining/ Total Time**

Replay and Simulation Tools User Interface (2/5)

Tool Path Reply Functionalities:

It allows you to visualize tool trajectory.

Tools for tool path replay:

- **Replay mode (a)**
- **Tool visualization mode (b)**
- **Tool path colors (c)**
- **Tracut display (d)**

Multi-Axis Flank Contouring.1

Tool animation — a

Replay positions
Start 1
Current 10
End 10

Feedrate =1000mm_mn
X=150.646mm Y=42.216mm Z=-45.223mm
I=-5.98869e-006 J=-0.0871484 K=0.996195
Machining time = 22s   Total time = 22s

OK

d

Display Tracut

b

Display all tool axes

c

Blue color for retract motions
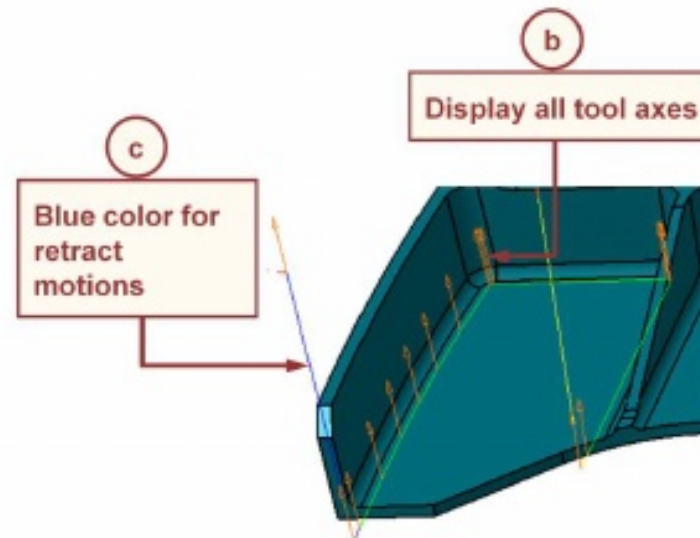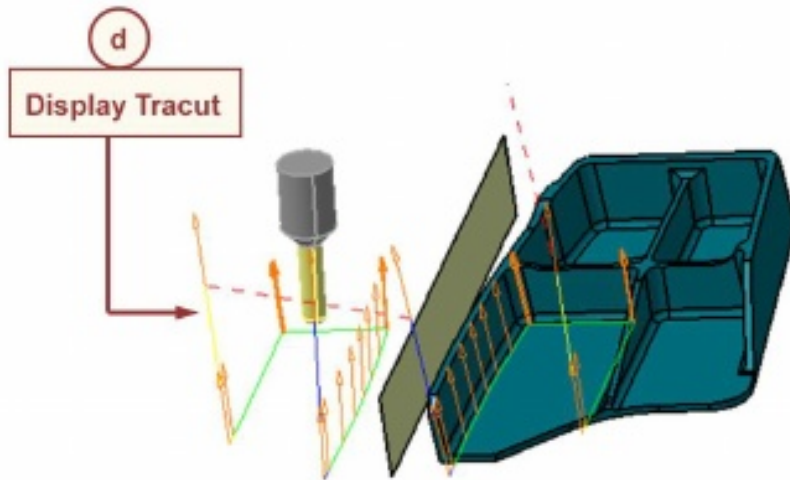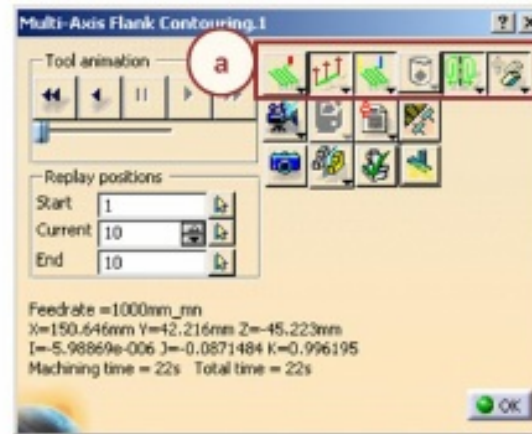
Replay and Simulation Tools User Interface (3/5)

Photo Functionalities:

- The Photo mode displays the result of the material removal at the end of the Machining Operation.
- This very fast simulation is based on a Pixel algorithm.
- The Photo simulation is performed in a new CATIA window called <u>Photo.</u>
- The result of this simulation can be analyzed to detect Gouging, Undercut and Tool Clash.
- Only available for 2.5 and 3 axis operation without Rotable.

Tools for photo simulation:

- Run photo simulation (1)
- Analyze tools available on the result of the photo
  - Using code color on gouge element or no remaining depth (2)
  - Using measure tool (3)
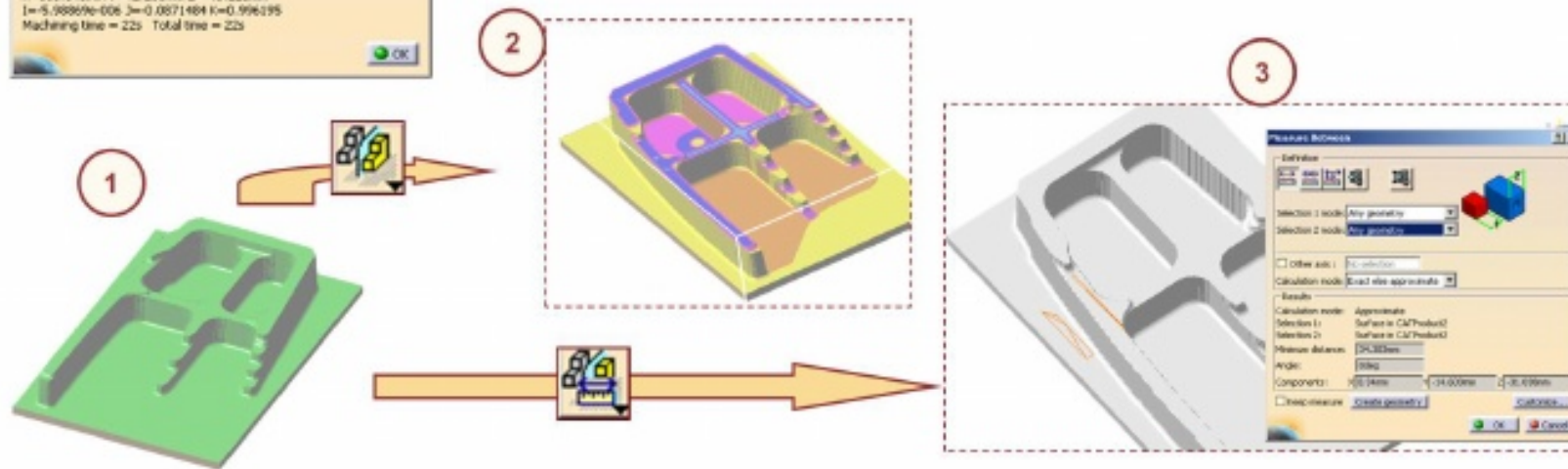
Replay and Simulation Tools User Interface (4/5)

Video Functionalities:

The Video mode is a material removal simulation. It gives an animation of the tool path and any Machine Rotations in the program are taken into account. The goal is to ensure that a good NC program will sent to the post processor.

Tools for Video simulation:

- **Run video simulation**
  - ◆ **Full video from beginning (1)**
- **Save video result**
  - ◆ **In a CATProduct / CGR file (externally) (2)**
  - ◆ **At the operation level (internally) (3)**
- **Run video simulation**
  - ◆ **From the last video result saved internally (4)**
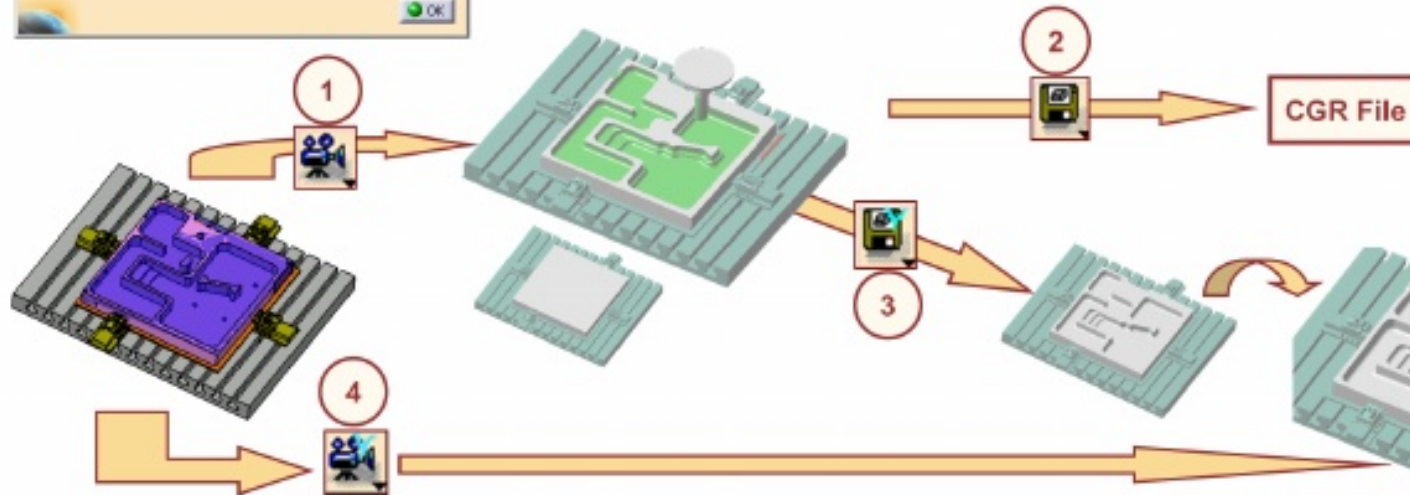
Replay and Simulation Tools User Interface (5/5)

Added Video Functionalities (need NVG license):

The goal is to analyze the result of the video simulation.

Tools for video simulation analyze:

- **Run video simulation (1)**
- **Save video result (at the operation level or externally)**
- **Analyze tools available on the result of the video**
    - ◆ **Using code color on gouge element or no remaining depth (2)**
    - ◆ **Using measure tool (3)**
- **Tool Collision analyze (4)**
- **Video Options**
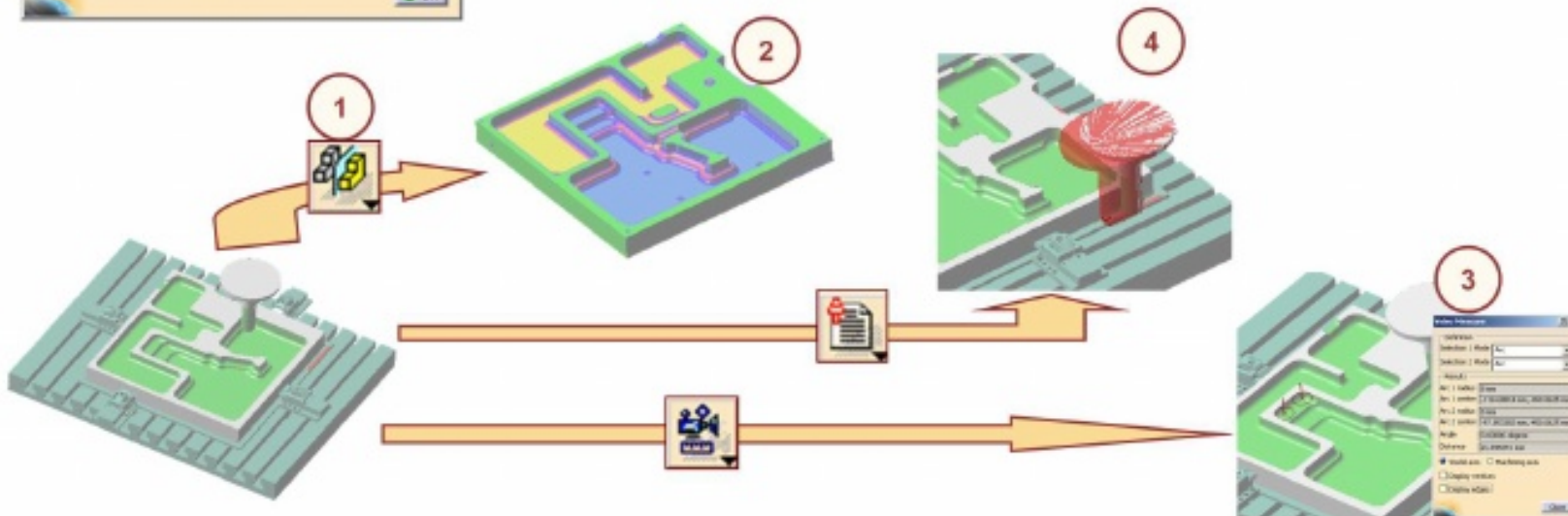
How to Replay a Tool Path (1/4)

The following **Replay** Options are available:

(A) (B) (C) (D) (E) (F)

A

Replay mode: Z Plane by Z plane

Until pp instruction

Continuous   Point by Point
Plane by Plane Feedrate by Feedrate

B

Tool Visualization Mode:

Tool at the last position | Tool displayed at each position

Tool Axis displayed at each position

C

Color Mode:

Color Mode

Same Color

The feedrate colors can
be define in the
tools/option/Machining/
general menu

D

Displays the tool point for the trajectory:

→ Displays the tool tip or center point and contact point (if it is stored) for tool trajectory

→ Displays the contact point (if it is stored) for tool trajectory otherwise, display the tool tip or center point

→ Displays the contact point only (if it is stored) for tool trajectory

→ Displays the tool tip or center point *(only possibility in 2.5 axis)*

E

Tracut Display Mode:

TRACUT instructions are taken into account to display tool path

TRACUT instructions are not taken into account to display tool path

F

Holder Visibility Options:

——— Hides the holder

Displays the holder

How to Replay a Tool Path (2/4)

A

B

C

A

Animation Stop
Backward   Forward

Go to Start   Display Complete
Tool Path

Animation Speed control for
continuous Replay Mode

Tool animation

Number of Points to be
visualized for Point by
Point Replay Mode

B

The Machining time and the Total time of the operation:
- Machining time: time corresponding to Machining + Finishing motions
- Total time: Machining time + time corresponding to approach, retract, Rapid (if value defined in the PO), delay

C

Replay positions
Start   1580
Current 1580
End   3245

Partial Tool Path Replay is possible by defining Start and End positions.

Start or End positions can be entered in text boxes or can be selected from 3D graphics area.

You can display the tool path of different operations together by selecting the machining operations with the CTRL key and click replay icon.

How to Replay a Tool Path (3/4)

Option for Tool Path Visualization:
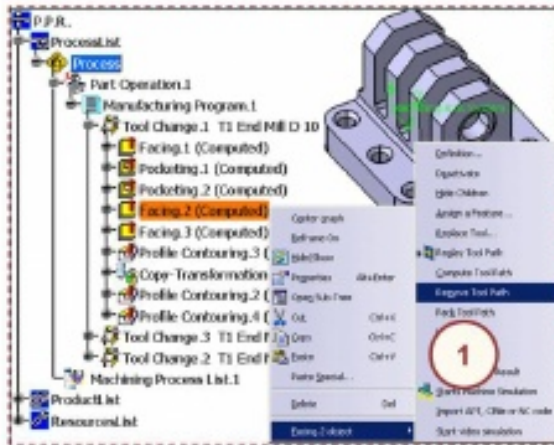
1 You can choose to store Tool path in a external file using:

MB3 a Machining Operation and Select **Pack Tool Path.**

2 The file is storing in the directory defined in Machining Setting or at the same location as the CATProcess

When a tool path is packed, the symbol appears

To store the tool path in the model, do the same process and click Unpack
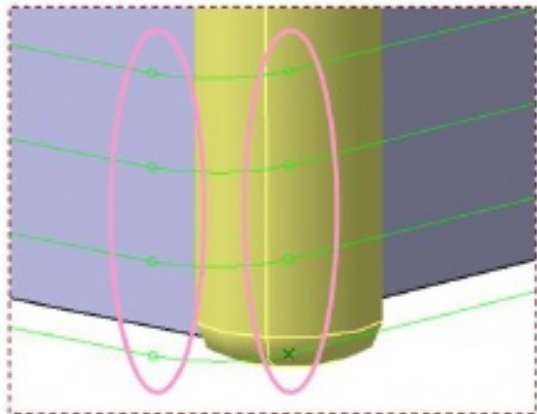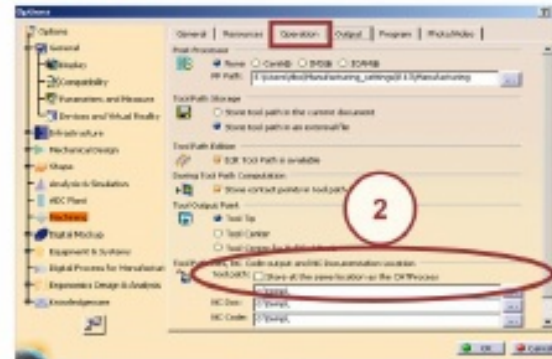
Display Circles during Replay

symbol « o » are displayed at Circle motion extremities

How to Replay a Tool Path (4/4)

Easier Navigation in NC Program:

Optionally, Computed Tool Paths can be displayed permanently.

- Controlled by a setting in Tools/Options.
- Operates on the Current Part Operation.
- Double Click on Tool path openes the Machining Operation Editor.
- Show/ No Show of Tool path for Current Machining Operation.
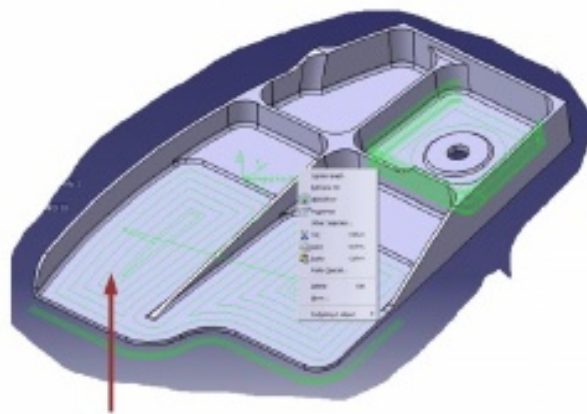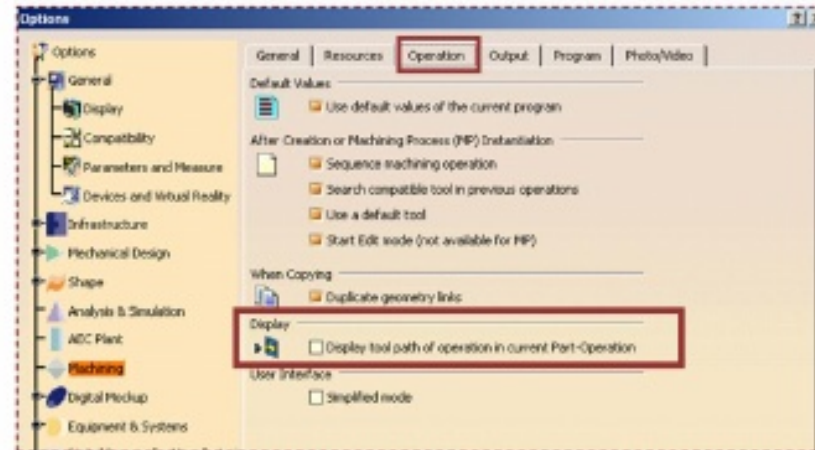
'Show' – Tool path
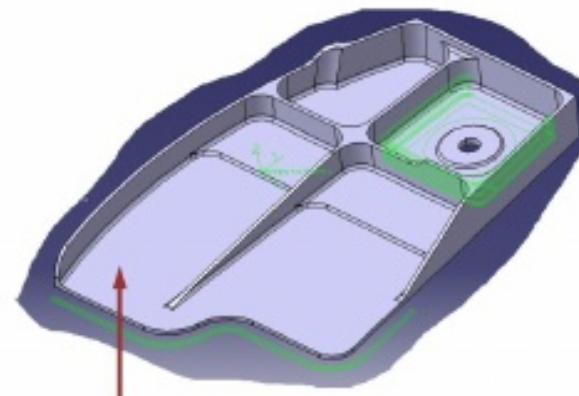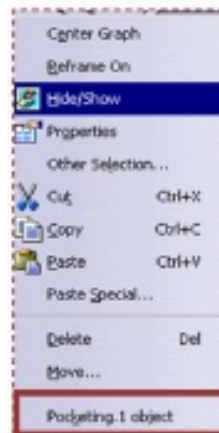
'Hide' – Tool path

Material Removal Simulation by Photo (1/4)

The **Photo** mode displays the result of the material removal at the end of the Machining Operation.

- This very fast simulation is based on a Pixel algorithm.
- The Photo simulation is performed in a new CATIA window called <u>Photo.</u>
- The result of this simulation can be analyzed to detect Gouging, Undercut and Tool Clash.
- Only available for 2.5 and 3 axis operation without Rotable.

1. **Select Manufacturing Program or Operation in the tree. Then Right-Click and select Replay Tool Path in the contextual menu**
   or
   Select the operation in the tree and click icon in the menu bar
   or
   Edit the Operation and use Replay button
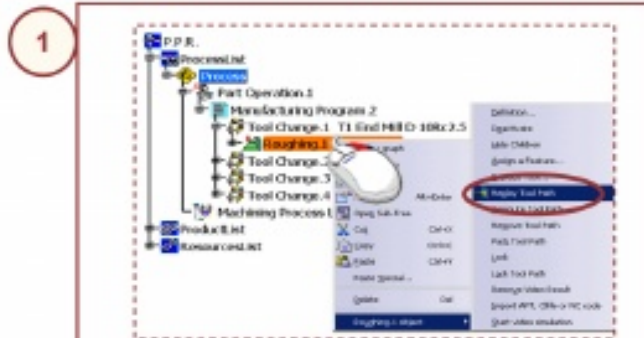
2. **Then click icon to start the Simulation**
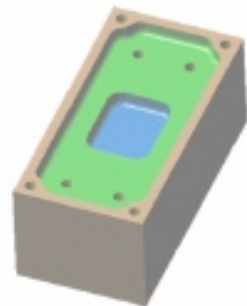
Photo

Material Removal Simulation by Photo (2/4)
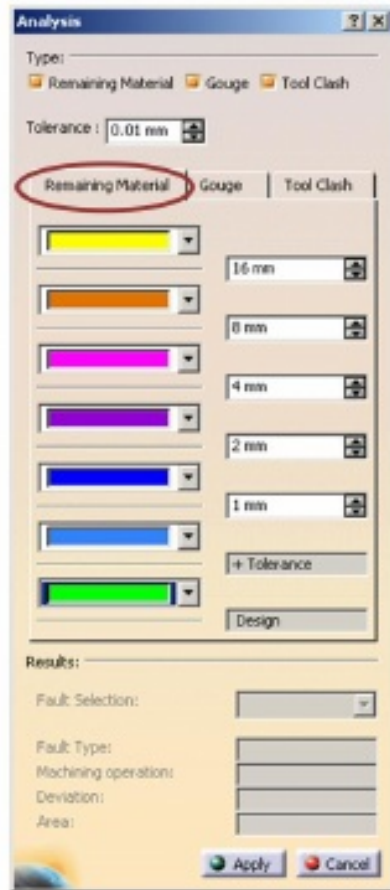
Analyze Photo to compare the machined part with the design part
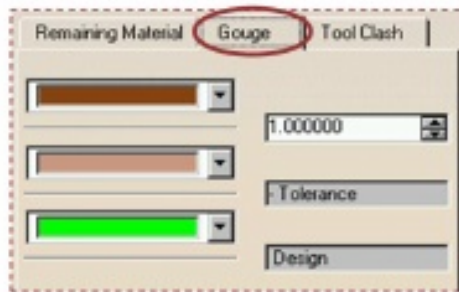
The following errors can be detected using the Analyze Photo capability:

- **Remaining Material (Undercut)**: Areas where the tool has left behind material on the work piece
- **Gouge**: Areas where the tool has removed excess material from the work piece
- **Tool Clash**: Areas where the tool collided with the work piece during a rapid move

These errors are determined according a user-defined **Tolerance**.

Results of the comparison are reflected on the work piece, based on the extent of severity of the fault and the customized color settings
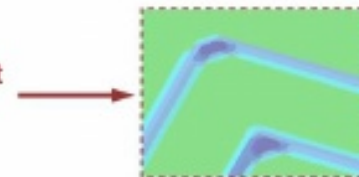
Material Removal Simulation by Photo (3/4)

1. **Select the type of Analysis you want (Remaining Material and/or Gouge and/or Tool Clash)**
2. **Define the tolerance**
3. **Specify the colors used to highlight the Areas within tolerance, Tool Clashes, Gouges and Undercuts**
4. **Specify the rate according to each color**
5. **You can now apply**

Tool Clash means:
- **Rapid motion in material**
- **Contact with the part of the tool which not cut**
- **Contact with the holder (if the option is tagged in tool Clash tab page)**

The list of detected faults are listed in the **Faults** combo box (Gouge, Undercut and Tool Clash) and detailed information related to these faults are displayed (Type, Machining Operation, Deviation and Area)

At any time you can pick on the surface of the work piece and a dialog box appears giving information about the picked point

- **The operation used for removing material.**
- **The normal deviation between the work piece and the design part.**
- **The X, Y, and Z coordinates of the pick point.**
- **The tool used for machining.**

Material Removal Simulation by Photo (4/4)

« Close up » Option allows you to improve the visualization of the analysis result

You can access to the « Close up » menu with MB3 on the photo or analysis window.
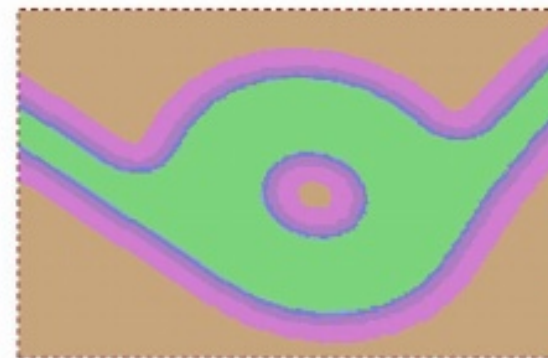
To use the « close up » :

- **Zoom on the interested zone**
- **Select « Close up » in the contextual menu**
- **Select Stock in the contextual menu to go back**

Other Selection...
Closeup
Stock
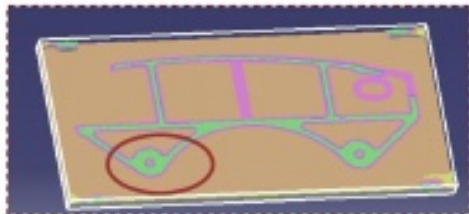
Zoom
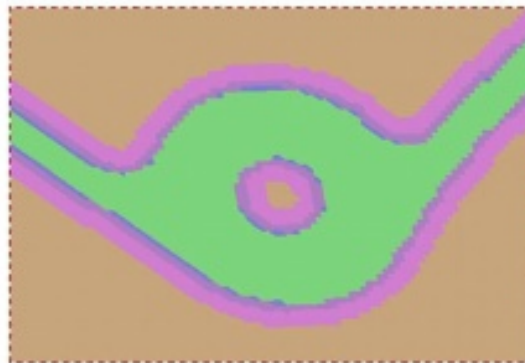
Closeup

Material Removal Simulation by Video (1/3)

The Video mode is a material removal simulation. It gives an animation of the tool path and any Machine Rotations in the program are taken into account. The goal is to ensure that a good NC program will sent to the post processor.

1. Select the Manufacturing Program or Operation in the tree. Select the Replay Tool Path option using the contextual menu
   or
   Select the Operation in the tree and click the Replay Tool Path icon in the toolbar
   or
   Edit the Operation and click the Replay button
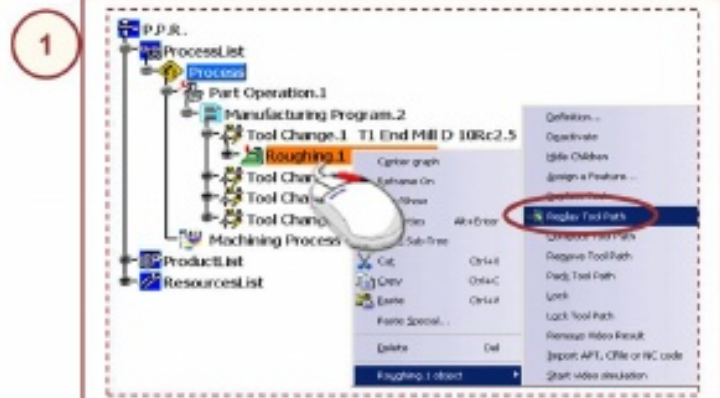
2. Click the icon to start the Simulation
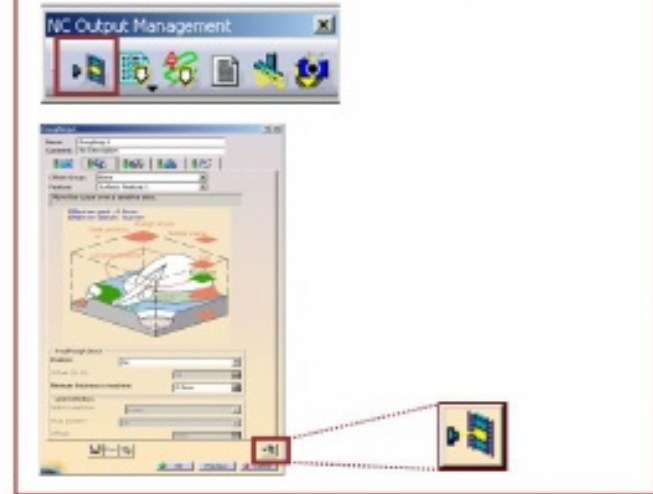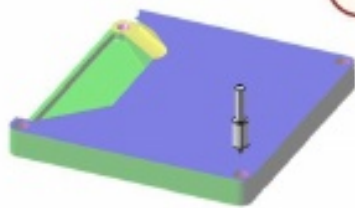
You can save the video simulation.

Material Removal Simulation by Video (2/3)
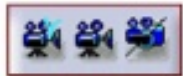
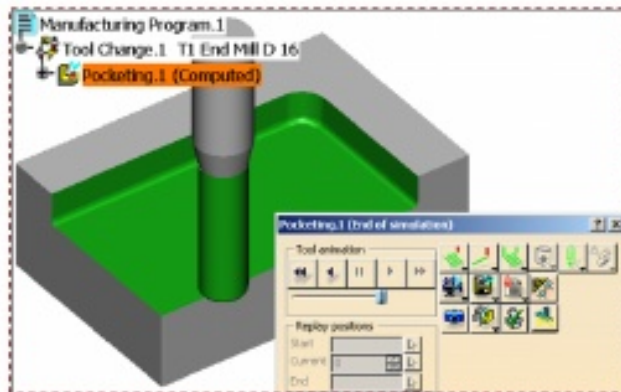**Simulation from operation's video result:** video simulation from saved result of the previous video.

Full video: video simulation for complete program or part operation (depending on setting)

Mixed Photo/Video: photo simulation is up to the operation just before the selected operation, then video simulation is done on the selected operation.
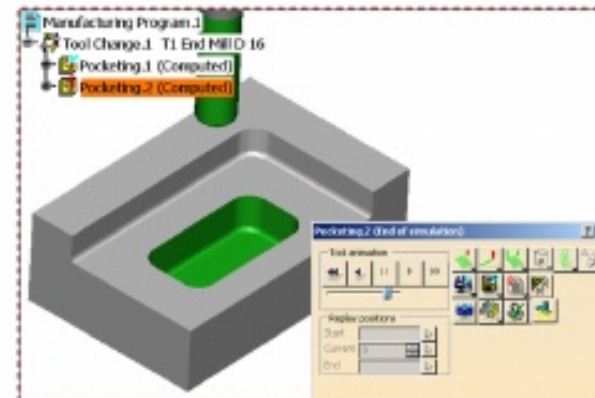
If the Replay mode is set to Point to Point, the number of points value is taken into account.

If the Replay mode is set to continuous, the slider position is taken into account for adjusting the speed of the animation

Material Removal Simulation by Video (3/3)

Save Result

Save video result as CATProduct/CGR: Video simulation result is saved in a CATProduct (imbedded WPC format file for better precision) +.CGR File (for representation). It can be reused in as Stock in Part operation or in SMG Roughing operation.

Associate video result to Machining operation: Save the result of the video in the operation. A material removal is displayed starting from the previous saved result.

Save video result as CGR: Video simulation result is saved under a .CGR File. It can be reused in as Stock in Part operation or in SMG Roughing operation.

Associate video result to Machining operation: Save the result of the video in the operation. A material removal is displayed starting from the previous saved result

You can store video result files (CATProduct) in the same folder as the CATProcess or at any other location.

Tools > Options > Machining > Output

Video result: ☐ Store at the same location as the CATProcess

C:\DOCUME~1\xhr\LOCALS~1\Temp\
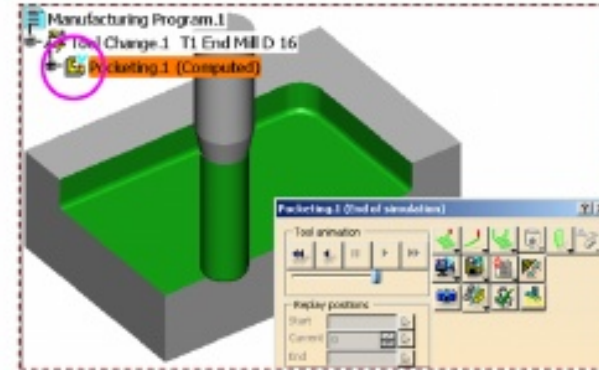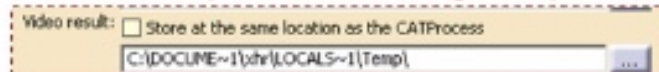
- The Video result may become incoherent if operations used in its creation are modified. Incoherent Video results must be removed by the user.

- The Remove Video Result contextual command allows you to remove a Video Result that is stored on an operation. An operation that has a stored Video result is indicated by a check mark in the tree.

- Video results are stored in the NC Code output directory.

NC Manufacturing Verification NVG (1/5)

This is an Advanced Tool Path Verification capabilities for multi-axis positioning as well as for multi-axis machining.

The accuracy of machined parts can be analyzed either by
- ◆ Detection and display gouges and remaining material
- ◆ Pick point analysis
- ◆ Measuring
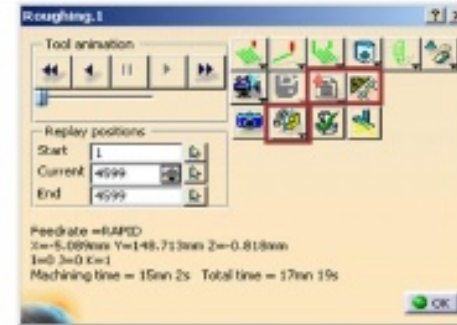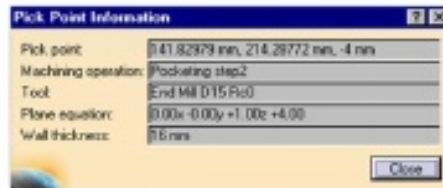
Collisions between the tool or tool holder and part or fixtures are detected and graphically visualized. The results of a material removal simulation can be stored in a reporting file
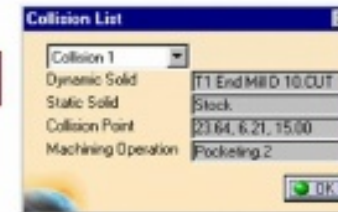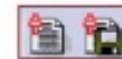
This icon allow you to access directly to the video option:
- ◆ Stop replay at each tool change
- ◆ Define the mode of collision detection
- ◆ Select the Video simulation in protected mode check box
  to continue the Video simulation by skipping any cuts that
  cause errors.

You can use the information regarding the clash point after the video window is killed in Replay dialog box. (You can access to collision points in Replay mode only if you have generated the collision report in Video mode before.)

NC Manufacturing Verification NVG (2/5)

Analyze Video enables you to compare the machined part with the design part

Functionality based on DMU development to make measure on you part during the simulation.

Measure ? X

Hide/Show

Measure Between two elements

Measure selected element

Measure Arc through 3 points

Same functionalities as in Photo mode:
The Differences is:
  ◆ **You can make analyze also for 4-5 axis and lathe operation.**
  ◆ **You can stop the simulation when you want then make a verification/analyze and continue the simulation**

Customize:
to display the information you want for a arc measure.

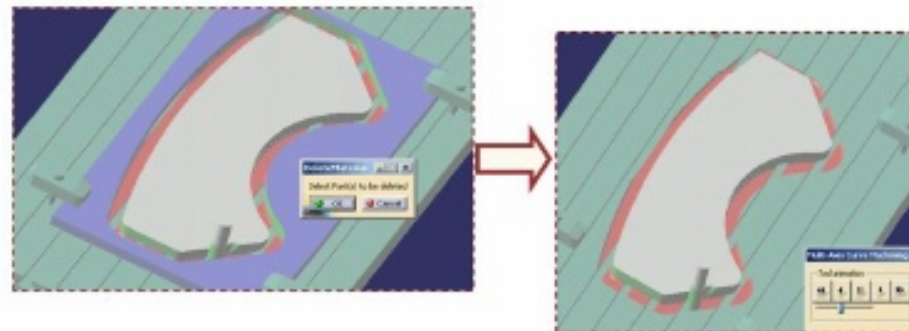# NC Manufacturing Verification NVG (3/5)

1 Measure in Video mode:

Allows to measure distance between 2 elements directly by selecting entity from Video simulation (no intermediate step). Different mode: between
- arc, plane, edge, point

Possibility to fix the axis measure according to the machining axis system.

2 Remove Chunks:

Allows to remove chunks for better collision detection and better CGR save.

Click the icon & select the part to remove & validate

NC Manufacturing Verification NVG (4/5)

Video Collision Report icon is to display a dialog box showing any collisions detected during the video simulation.

You Can Generate a Report txt File of collisions
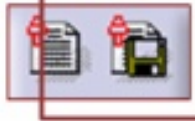
Collision concerns rapid motion in the material and contact between non- cutting tool part and material. List of the detected collision
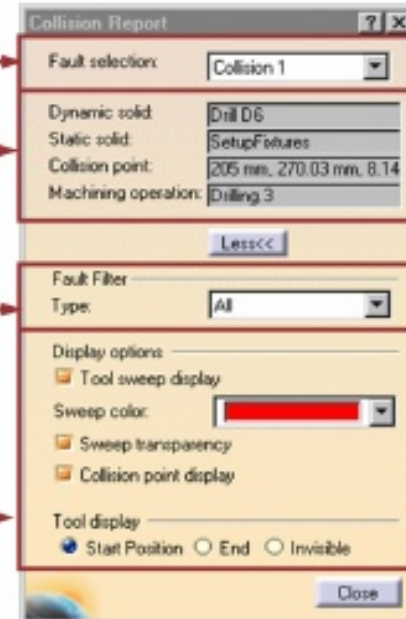
Collision list

Collision information:
- Elements in collision
- Coordinate of collision point

Collision filter:

Collision/ Tool Visualization option

Note that:
The Collision Condition setting must be set to continue through in the replay dialog box or in Tools > Options. You can choose between:
 ◆ **No report (Ignore)**
 ◆ **Stop video replay at each collision**
 ◆ **Detect collision but don't stop the replay**
You can define: touch is collision if the tool holder is taken into account during collision checking.
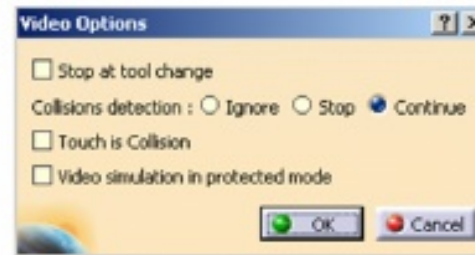
NC Manufacturing Verification NVG (5/5)

Load Simulation Result:
You can load the associated Video results for analysis in an analysis window.

The 'Load Simulation Result' command is available in contextual menu of a machining operation which has an associated Video Result CATProduct.

This command opens the Video Result CATProduct in a Video window along with 'Analysis' toolbar for analysis of the machined stock, collision results, and so on.

Video Analysis commands:

Video measure: Measures distance between 2 elements directly by selecting entity from Video simulation.

Analysis: Compares the machined part with the design part.

Remove Chunks: Allows to remove chunks for better collision detection and better CGR save (Stock must be split).

Save video result as CATProduct/CGR

Collision List:

Tools for Optimization

This lesson covers the following topics:
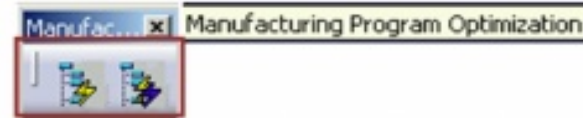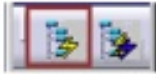
- Auto Sequence
- Auxiliary Operations

Auto Sequence

You will learn how to Sequence the Machining operations Automatically.

- **General Process for Auto Sequencing**
- **Administrator level**
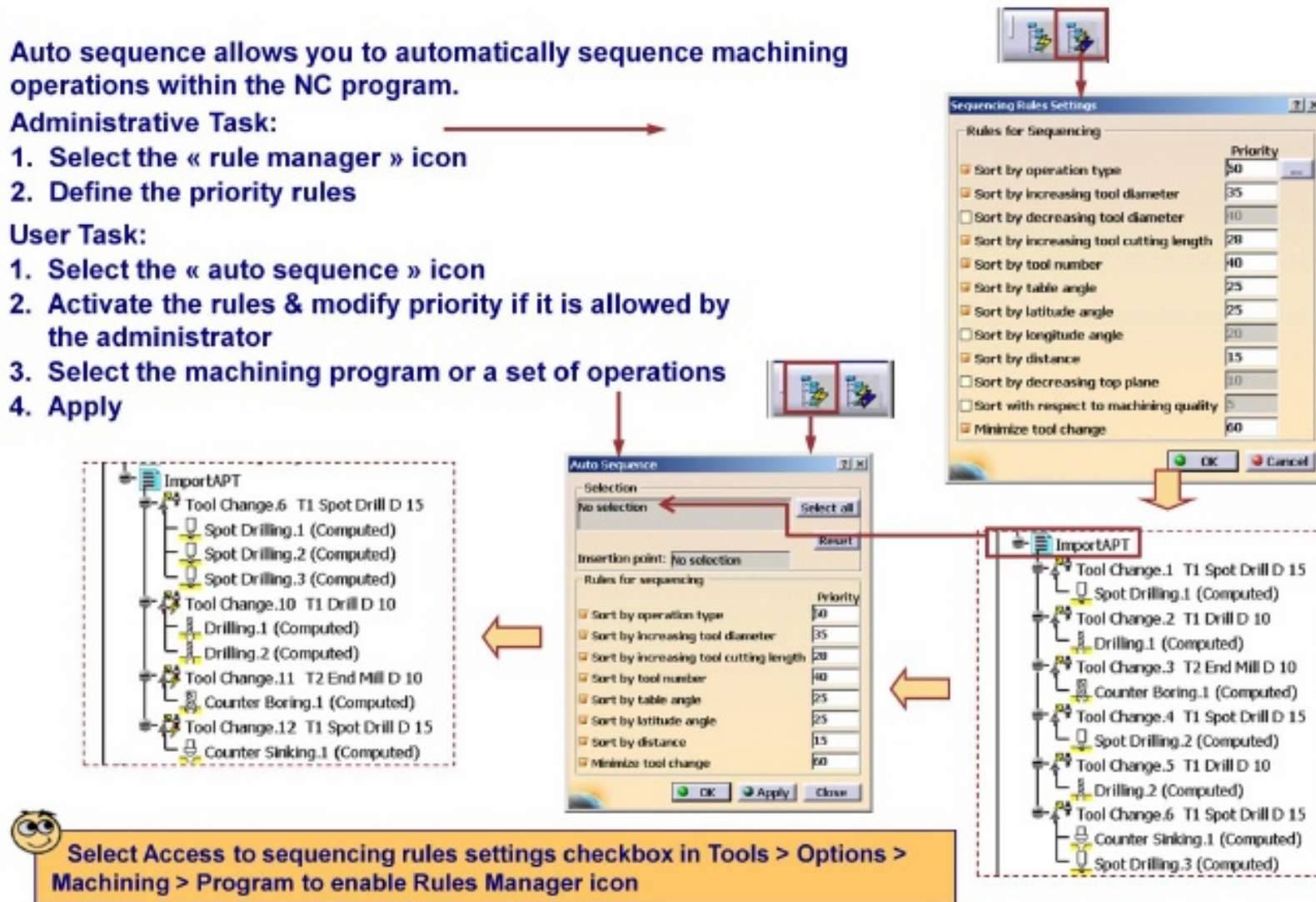- **User level**

General Process for Auto Sequencing

Auto sequence allows you to automatically sequence machining operations within the NC program.

Administrative Task:

1. **Select the « rule manager » icon**
2. **Define the priority rules**

User Task:

1. **Select the « auto sequence » icon**
2. **Activate the rules & modify priority if it is allowed by the administrator**
3. **Select the machining program or a set of operations**
4. **Apply**

Sequencing Rules Settings

Rules for Sequencing

	Priority
☑ Sort by operation type	50
☑ Sort by increasing tool diameter	35
☐ Sort by decreasing tool diameter	40
☑ Sort by increasing tool cutting length	28
☑ Sort by tool number	40
☑ Sort by table angle	25
☑ Sort by latitude angle	25
☐ Sort by longitude angle	20
☑ Sort by distance	15
☐ Sort by decreasing top plane	10
☐ Sort with respect to machining quality	5
☑ Minimize tool change	60

OK   Cancel

ImportAPT
- Tool Change.6 T1 Spot Drill D 15
  - Spot Drilling.1 (Computed)
  - Spot Drilling.2 (Computed)
  - Spot Drilling.3 (Computed)
- Tool Change.10 T1 Drill D 10
  - Drilling.1 (Computed)
  - Drilling.2 (Computed)
- Tool Change.11 T2 End Mill D 10
  - Counter Boring.1 (Computed)
- Tool Change.12 T1 Spot Drill D 15
  - Counter Sinking.1 (Computed)

Auto Sequence

Selection

No selection   Select all

Reset

Insertion point: No selection

Rules for sequencing

	Priority
☑ Sort by operation type	50
☑ Sort by increasing tool diameter	35
☑ Sort by increasing tool cutting length	20
☑ Sort by tool number	40
☑ Sort by table angle	25
☑ Sort by latitude angle	25
☑ Sort by distance	15
☑ Minimize tool change	60

OK   Apply   Close

ImportAPT
- Tool Change.1 T1 Spot Drill D 15
  - Spot Drilling.1 (Computed)
- Tool Change.2 T1 Drill D 10
  - Drilling.1 (Computed)
- Tool Change.3 T2 End Mill D 10
  - Counter Boring.1 (Computed)
- Tool Change.4 T1 Spot Drill D 15
  - Spot Drilling.2 (Computed)
- Tool Change.5 T1 Drill D 10
  - Drilling.2 (Computed)
- Tool Change.6 T1 Spot Drill D 15
  - Counter Sinking.1 (Computed)
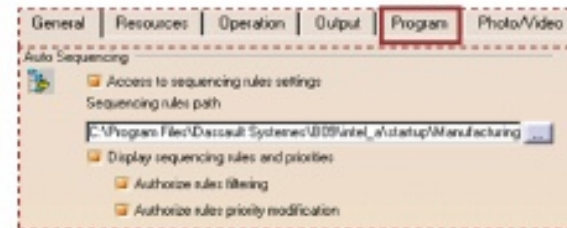 - Spot Drilling.3 (Computed)

Select Access to sequencing rules settings checkbox in Tools > Options > Machining > Program to enable Rules Manager icon
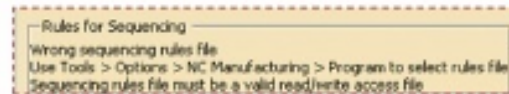
Auto Sequence - Administrator Level
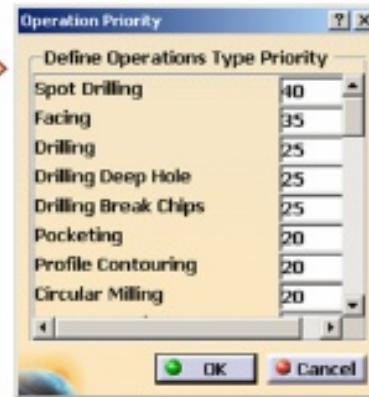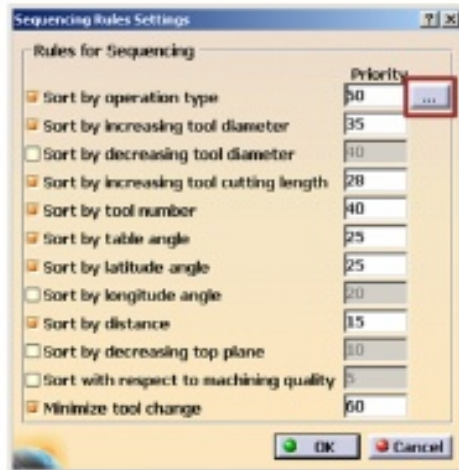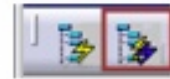
Sequencing Rules Settings:
- Check the rules you want to use
- Affect to each of then a priority (0: lowest priority) button to access to Operation priority

Sequencing Rules Path: Path where the file containing Sequencing rules is stored.
Make sure that the document in the sequencing rules path (AllSequencingRules.CATProduct in the example above) is accessible in Read-Write.

The settings in the Auto-Sequencing area are mainly intended for the administrator

Access to sequencing rules settings: Select the first check box to authorize user access to sequencing rules

Display sequencing rules and priorities: Select the second check box to authorize the display of sequencing rules and priorities in the user's view. In this case two more check boxes can be selected in order to:
- Allow the user to filter rules
- Allow the user to modify rule priorities

Auto Sequence – User level
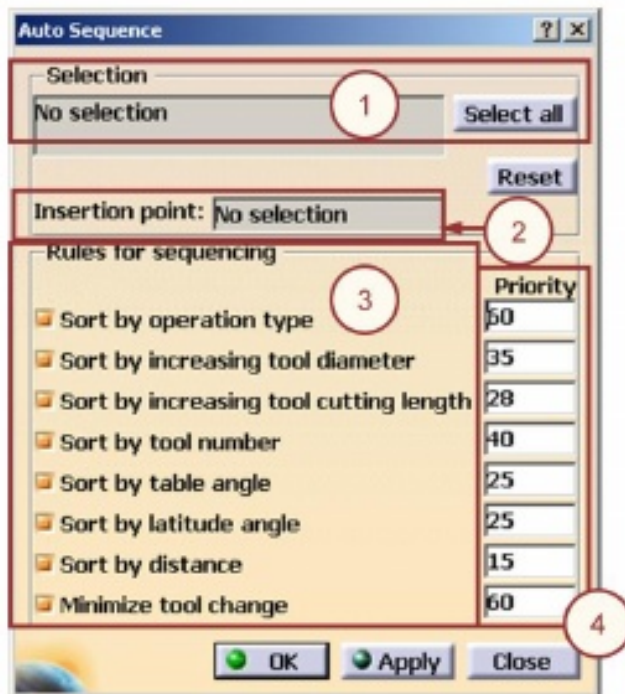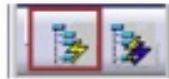
Auto Sequence:

1. **Select all or a set of operations on a program**
   - ◆ **Manually in the tree or**
   - ◆ **Press Select all**
   - ◆ **Click reset to remove all the selected operations**

2. **Select the level of insertion of the ordered operation**

3. **Select the Rules provided by the administrator**

4. **Modify the priority if it's allowed by the administrator**

Auxiliary Operations

You will learn how to create an Auxiliary Operation.

- Need of Auxiliary Operations
- Creating manually an Auxiliary Operation
- Creating Auxiliary Operation Automatically
- Auto Complete
- More Details about Auto Complete
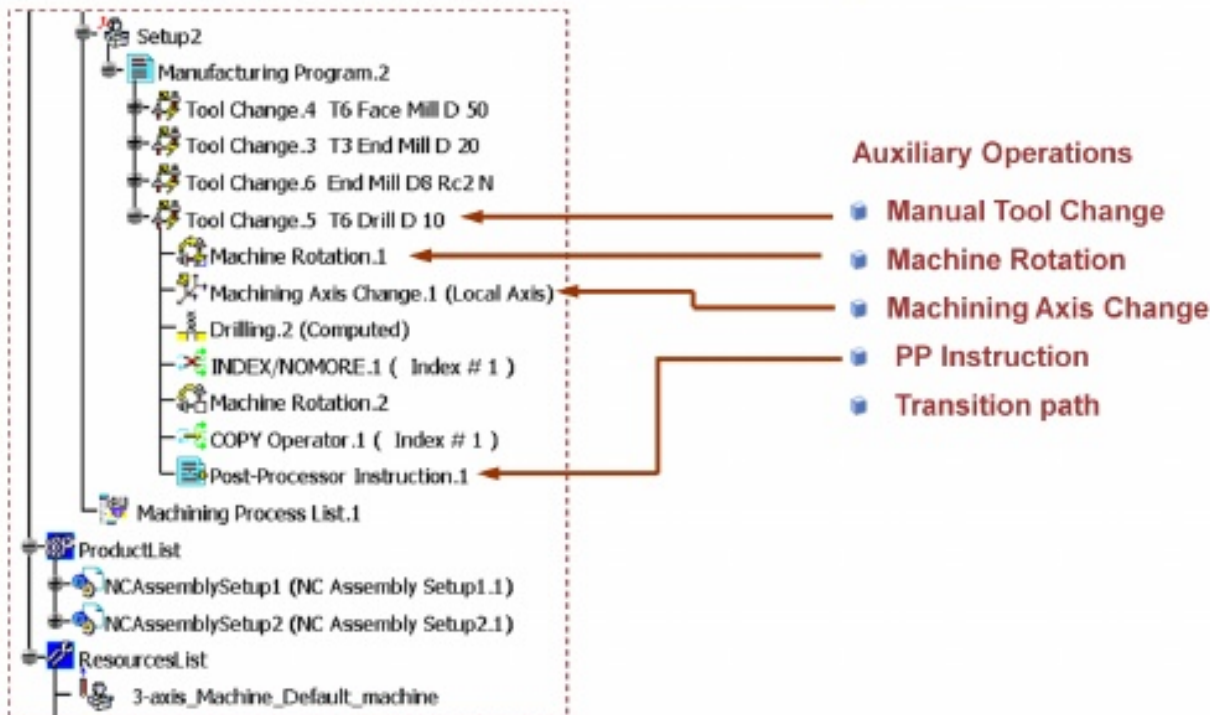- Creating manually a Copy Operation

Why Do You Need Auxiliary Operations

An Auxiliary Operation is a control function such as Tool Change, Machine Table/Head Rotation or a single PP Instruction. These operations may be interpreted by a specific Post-processor.
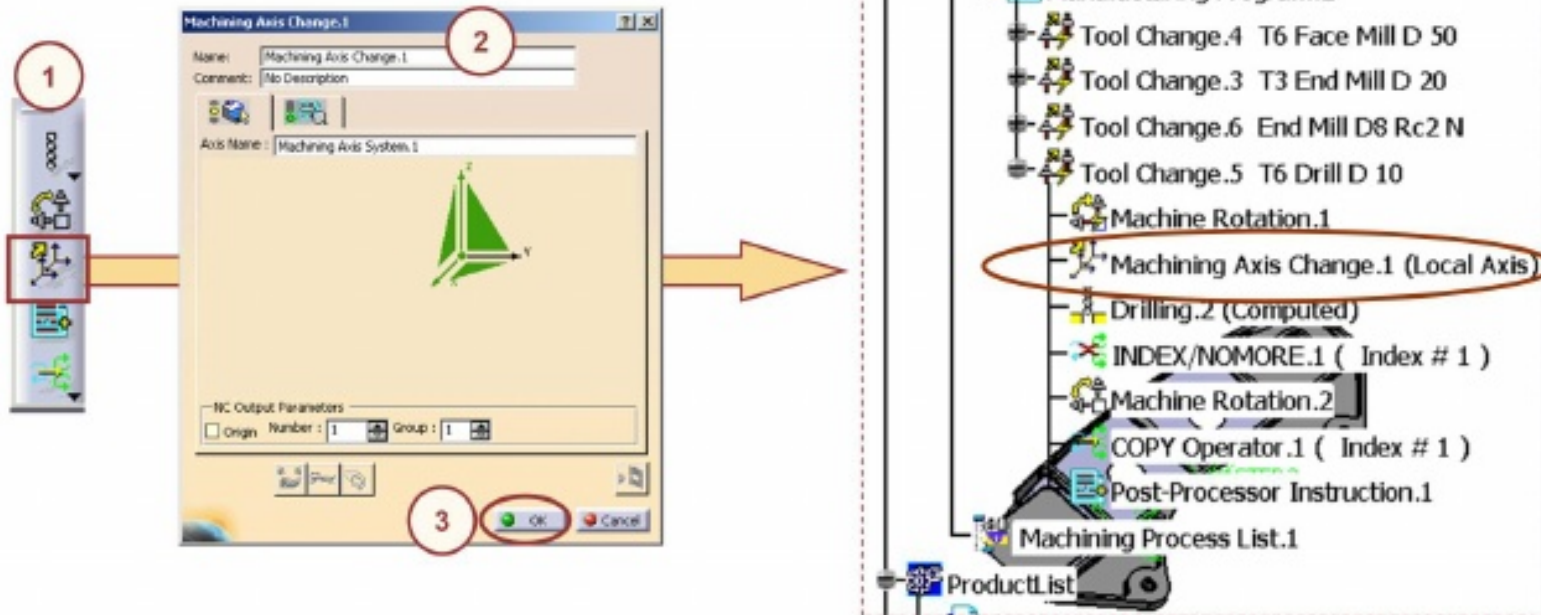
About Auxiliary Operations:
- Auxiliary Operations are predefined syntaxes stored in the Post-processor Table.(PP Table)
- The PP Table is referenced by the PO's Machine-Tool.
- All the syntaxes in the PP Table are customizable by the user.

Setup2
- Manufacturing Program.2
  - Tool Change.4 T6 Face Mill D 50
  - Tool Change.3 T3 End Mill D 20
  - Tool Change.6 End Mill D8 Rc2 N
  - Tool Change.5 T6 Drill D 10
  - Machine Rotation.1
  - Machining Axis Change.1 (Local Axis)
  - Drilling.2 (Computed)
  - INDEX/NOMORE.1 (Index # 1)
  - Machine Rotation.2
  - COPY Operator.1 (Index # 1)
  - Post-Processor Instruction.1
- Machining Process List.1
- ProductList
  - NCAssemblySetup1 (NC Assembly Setup1.1)
  - NCAssemblySetup2 (NC Assembly Setup2.1)
- ResourceList
  - 3-axis_Machine_Default_machine

Auxiliary Operations
- Manual Tool Change
- Machine Rotation
- Machining Axis Change
- PP Instruction
- Transition path

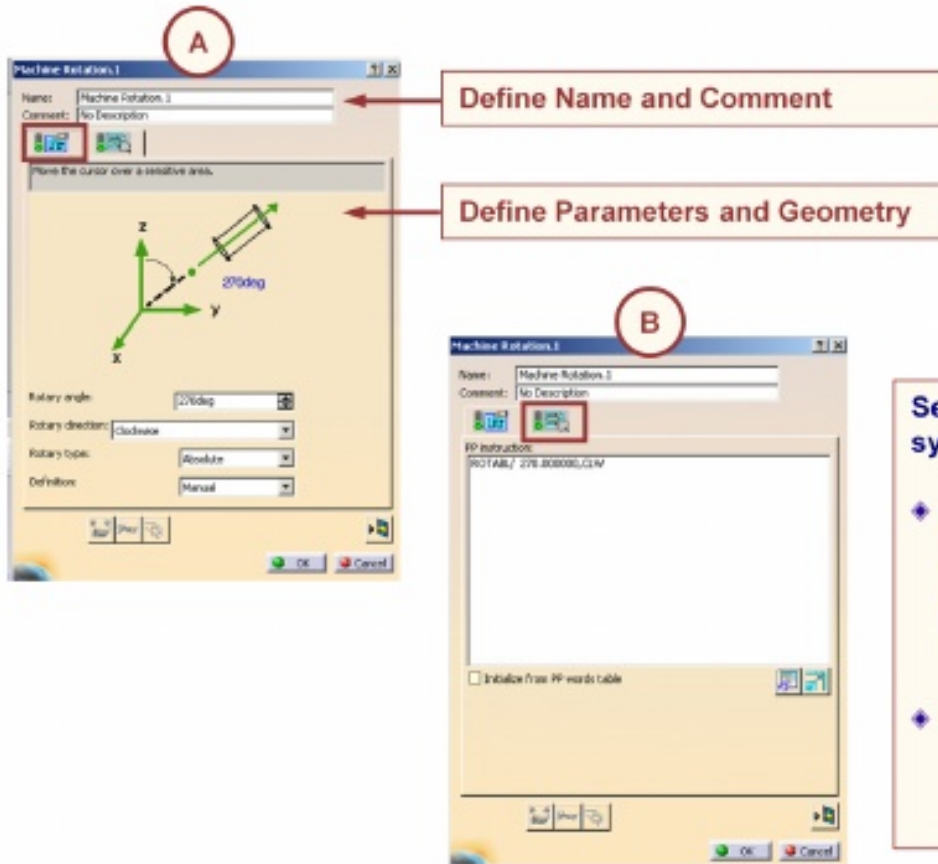# Creating Manually an Auxiliary Operation (1/2)

1. **Click Auxiliary Operation Icon**

2. **The new Operation is created after the current one**

   The Operation dialog box appears to edit it.

3. **Confirm Operation creation**

Creating Manually an Auxiliary Operation (2/2)

The Auxiliary Operation dialog box is composed of two parts:
- **Parameter/ geometry definition (A)**
- **PP syntax (B)**

(A)

Define Name and Comment

Define Parameters and Geometry

(B)

Select the Syntax Tab Page to display the syntaxes associated to the Operation

- ◆ **Initialize From PP words table:**
  The predefined syntax is read on the PPWords Table linked to the machine and the syntax parameters are updated with the Tool Change parameters

- ◆ **Otherwise:**
  Key your own user-syntax that will have no link with the PPTable

Creating Auxiliary Operations Automatically

1. **MB3 on the Manufacturing Program**
2. **Select the type of Auxiliary Operation to be automatically generated among:**
   - ◆ **Tool Changes**
   - ◆ **Machine Rotations**
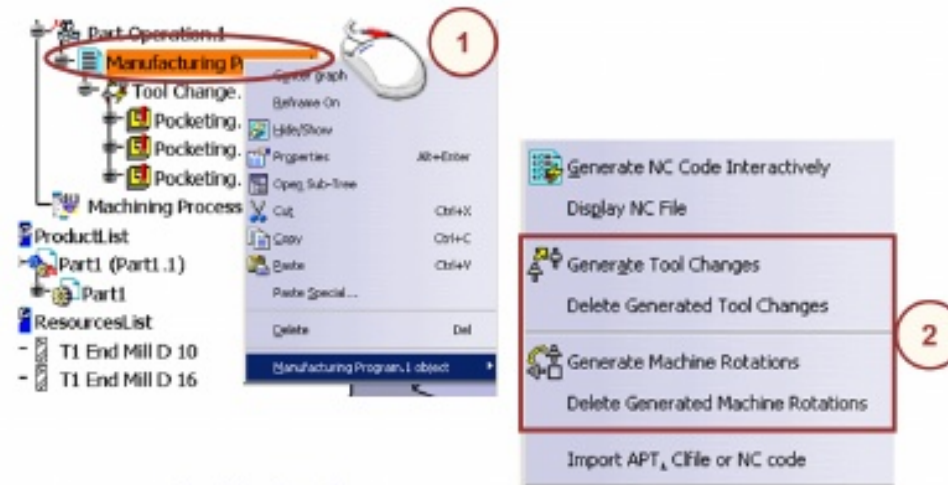
CATIA will check that all operations tool axes are reachable by the Part Operation's machine-tool and an Information Message or Warning Message will be displayed.
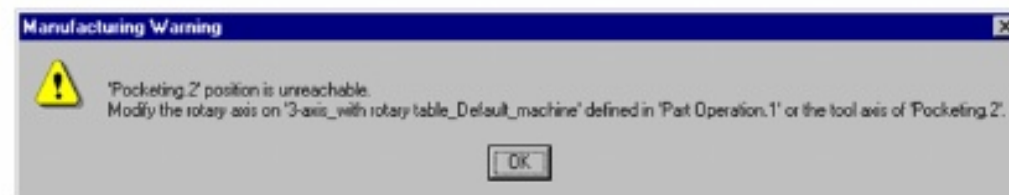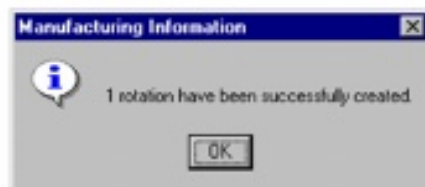
> It is also possible to remove all the Tool Changes and / or Machine Rotations by this automatic step.

What is Auto Complete

Auto Complete allows you to insert automatically transition path according to your machine tool and Transition/ Rotation planes defined in the PO.

IMPORTANT:
You need to define a Machine tool.
If you don't have a virtual machine tool you just insert standard rotable.

1. **Associate a Machine tool**
2. **Put the part in position**
3. **Define limit planes (traverse box, rotary planes) in PO**
4. **Select the automatic complete icon**
5. **Define your option for your transition motions**
6. **Run**

and

More Details about Auto Complete

Transition Pa...

A

B

C

Transition Paths

Selection

No selection

No program selected

Reset   Select All

1

2 Transition Path Creation

☐ To machining operation   ☐ To tool change

3 Machine Rotation Creation

☐ Table   ☐ Head

4 Priority Order

● Tool change   ○ Machine rotation

5 Retract/Approach Motion

○ Perpendicular to transition plane

● Along operation tool axis

6 Traverse Motion on Top

☐ Force motion on top plane of box

7 If No Transition Plane

☐ Compute axial-radial motion

8 At the end of the program...

☐ Return to the machine home position

● OK   ● Apply   Close

Icon details:

A. Generate automatically transition paths according the machine and the limit planes

B. Remove the transition paths generated

C. Modify transition path parameters

Note: You can use the first icon to recreate the transition. In this case the system remove all the transition paths and create them again

In fact this icon is here to not disturb V4 users

Dialog-box details:

1. Select the machining operations, or the program or All

2. Define Priority order
   - Tag tool change: then you will generate machine rotation before the Tool Change
   - Tag Machine rotation: then you will generate machine rotation before the tool change

3. Activate option to generate rotable or/and rothead

4. Define the transition path you want to generate:
   - Between machining operations
   - For tool changes
   - For machine rotation

5. Define the Approach and retract macro motions:
   - Perpendicular to transition plane (that you have defined in the Part operation)
   - Along operation tool axis up to the transition plane (that you have defined in the Part operation)

6. Force motions on top plane of Traverse box

7. If no transition plane has been defined, activate option to just create axial/radial motion

8. To generate additional Transition Path after the last MO to return to home.
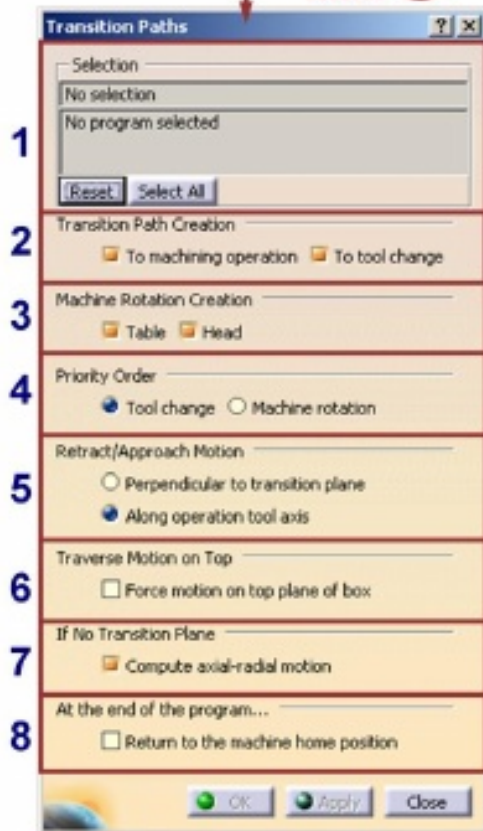
Creating Manually a Copy Operation

In case of identical or similar features, to minimize the number of operations,you can use copy operations. You have 3 kind of copy operations as given below:

Copy instructions: Allows to copy a sequence of operation including Tool Change & PP instruction (like a loop)
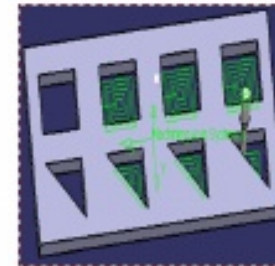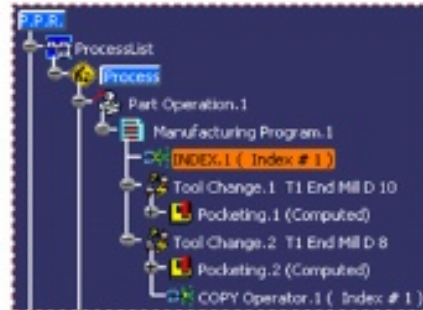
Tracut instructions: Allows to modify the trajectory of an operation by applying a transformation on it (no duplication)

Without Tracut

With Tracut
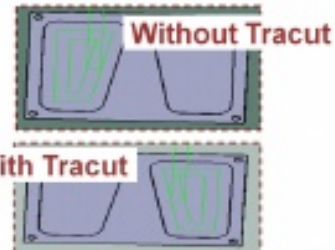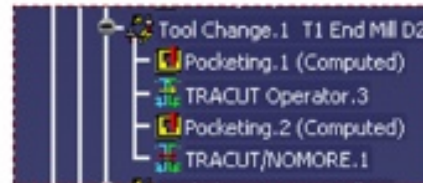
Copy transformation: Allow to duplicate tool path by applying a transformation on it (available only for machining operation using the same tool)
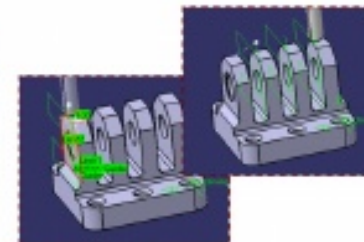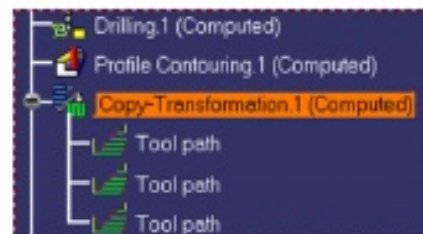
Auxiliary Operations- More Details

You will learn More Details about an Auxiliary Operation.

- Insert a Tool change
- Insert a Machining Rotation Operation
- Insert a Local Machining Axis
- Insert a Post- processor Instruction
- Copy Instruction and Transformation Management
- Tracut Instruction
- Auxiliary Operations Status
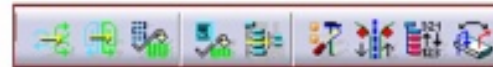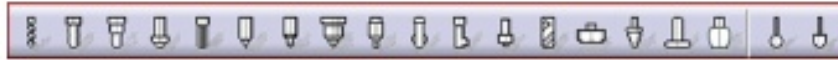
Insert a Tool Change Operation: General Process (1/2)

1 Select in the displayed list the Tool type to be created

2 Type the Name of the Tool Change Operation and a line of comment (Optional)

3 Select the Tool Tab Page to define your tool

4 Specify a name, a comment or a tool number that does not already exist to create a new tool

5 Use the 2D Viewer to modify the parameters of the tool. The 2D Viewer is updated with the new values
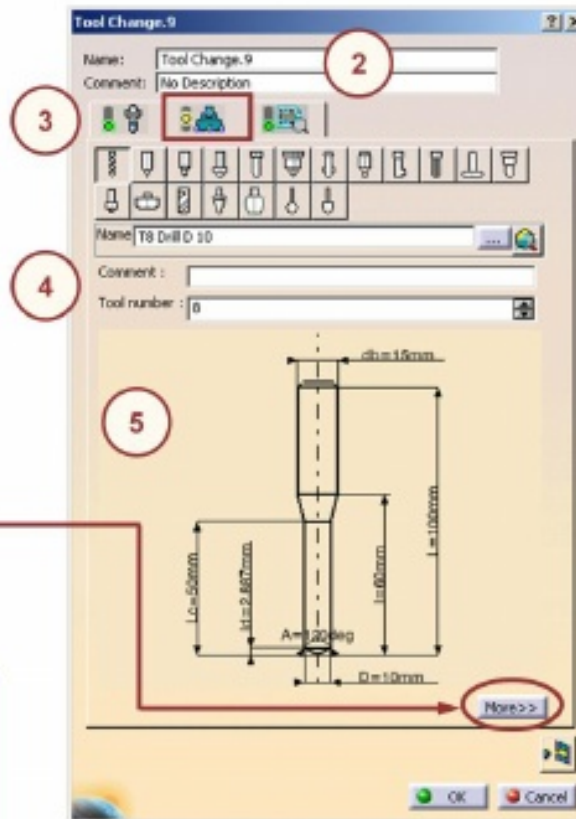
Click More>> to expand the dialog box to access all tool parameters such as Geometry, Technology, Feeds & Speeds and Compensation

For the following capabilities:
- Create a new tool
- Select an already existing tool from the current document
- Select another tool in a catalog by means of a query
(Refer lesson on 'Manage the tool of an Operation')

Insert a Tool Change Operation: General Process (2/2)

(6) **Select the Syntax Tab Page to display the syntaxes associated to the Tool Change operation**

Initialize From PP words table: the predefined syntax is read on the PPWords Table linked to the machine and the syntax parameters are updated with the Tool Change parameters

**Otherwise:** Key your own user-syntax that will have no link with the PPTable
♦ Use icon to refresh the syntax
♦ Use icon to maximize the text zone

Tool Change.1

Name: Tool Change.1
Comment: No Description

Sequence number: 1

PP instruction:
TOOLNO/3, 10.000000
TPRINT/T3 Drill D 10
LOADTL/3

☐ Initialize from PP words table **(6)**
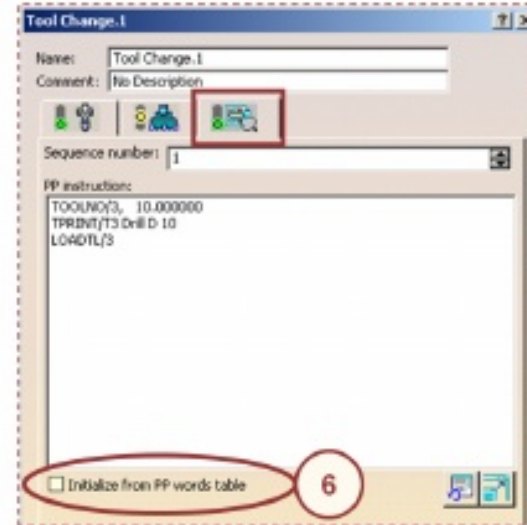
The Sequence Number allows you to choose one syntax associated to this command if several are defined in the PP Words Table.

Insert a Machining Rotation Operation: General Process

1 **Type the Name of the Machine Rotation Operation and a line of comment (Optional)**

2 **Define Rotation Parameters in the Properties Tab Page**
- **Rotary Angle** in Degrees
- **Rotary Direction** between CLW, CCLW or Both (Shortest)
- **Rotary Type** Absolute

Define the associated PPWords Syntax in the Syntax Tab Page

3 **Initialize From PPTable**: the predefined syntax is read on the PPWords Table linked to the machine and the syntax parameters are updated with the Rotation parameters

Otherwise: Key your own user-syntax that will have no link with the PPTable
- **Use icon to refresh the syntax**
- **Use icon to maximize the text zone**

4 **Simulate the Rotation by selecting the Replay button**
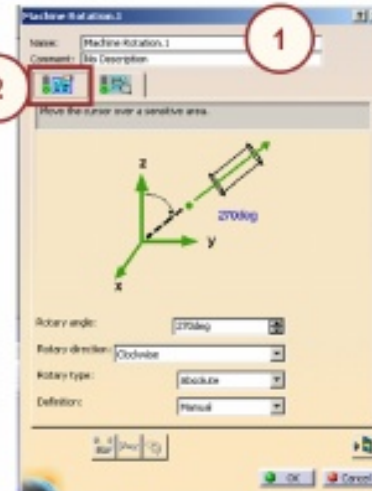
> The Machine Rotation operation can be generated only if a machine- tool with table rotation has been selected on the Part Operation. The rotation axis (A,B or C) is read on the machine-tool.

About the Head Change Activity

You can insert a new Head Change activity in the program, if an NC machine having at least one head is assigned to the Part Operation.

When you add a Head change activity, a new Tool change activity will also be added just after that Head change Activity, if the Next activity is not the Tool change.

- P.P.R.
  - ProcessList
    - Process
      - Part Operation.1
        - Manufacturing Program.1
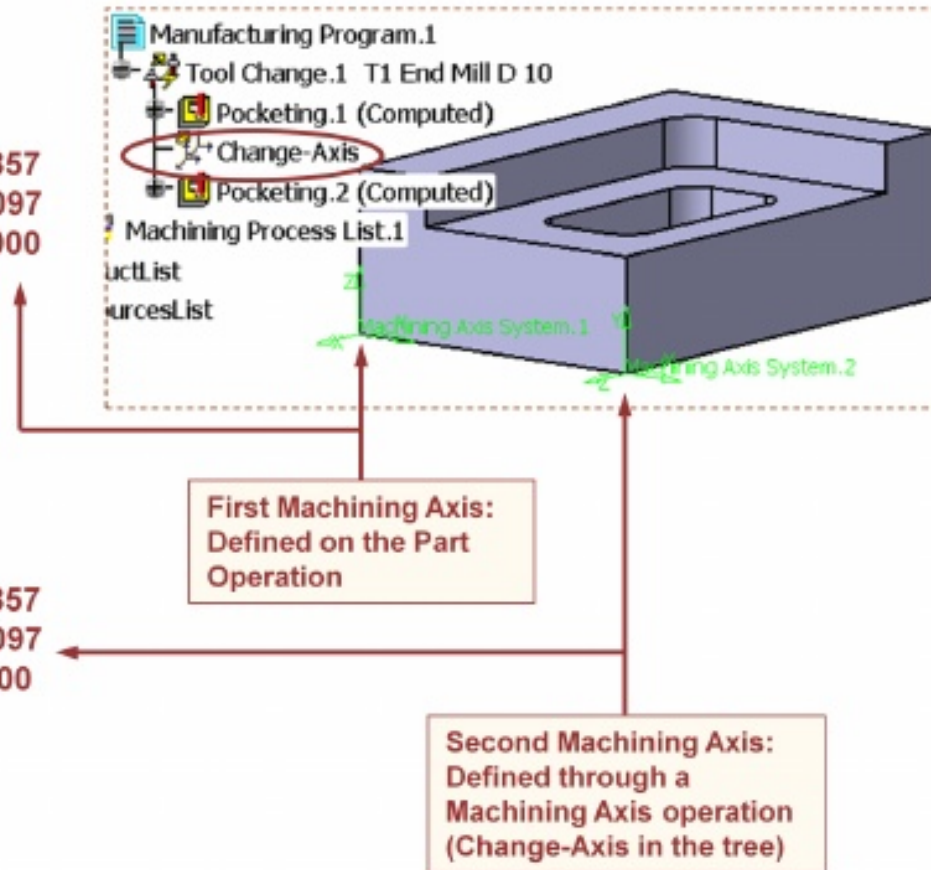   - Machine Instruction.1
          - Tool Change.4 T1 End Mill D 10
          - HeadChange.11
          - Tool Change.5 T1 End Mill D 10

If the Head change is the first activity which you are adding in the Manufacturing Program, then no Tool Change activity will be created after the Head Change Activity.

- P.P.R.
  - ProcessList
    - Process
      - Part Operation.1
        - Manufacturing Program.1
          - HeadChange.12
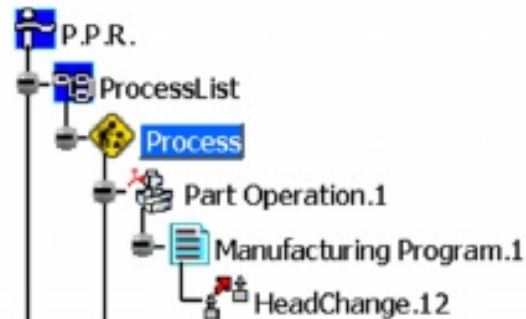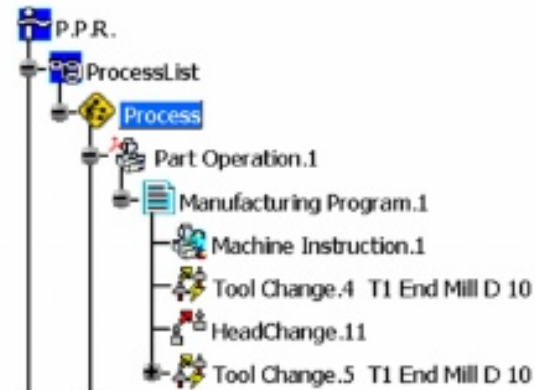
The Head change command is available with MSG.prd license.

Insert a Head Change Activity: General Process

1   Type the Name of the Head Change and a line of comment (Optional)

2   Define a head from the Interchangeable Head List in the Properties tab.
The Interchangeable Head List proposes all the available heads on the machine that is assigned to the Part Operation.

3   Define the associated PPWords Syntax in the Syntax Tab Page
Initialize from PP words table: the predefined syntax is read on the PPWords Table linked to the machine.

Otherwise: Enter your instruction for the head change that will have no link with the PPTable.

4   Click OK to create the head change in the program.

HeadChange.1

Name: HeadChange.1   **1**
Comment: No Description

Head Change
Interchangable Head List : Tete V   **2**

HeadChange.1

Name: HeadChange.1
Comment: No Description

PP instruction:
NC_HEAD_CHANGE/Tete V

3

Initialize from PP words table

4   OK   Cancel

The machine assigned on the part operation MUST have at least one head to create a head change activity.

Insert a Local Machining Axis: General Process (1/2)

1 Type the Name of the Machining Axis Operation and a line of comment (Optional)

2 Define the new Machining Axis in the Properties ⬛ Tab Page

- ◆ Click the symbol representing the origin in the sensitive icon and select a point or a vertex to fix the machining axis origin
- ◆ Select the axes in the sensitive icon to specify their orientation
- ◆ Key an Axis Name which is displayed in CATIA screen

> **Activation of the Origin Check box, with a number and a group, will generate in the output the following syntax:**
> **$$ ORIGIN/ X, Y, Z, Number, Group**

3 Define the associated PPWords Syntax in the Syntax ⬛ Tab Page

Initialize From PPTable: the predefined syntax is read on the PPWords Table

Otherwise: Key your own user-syntax that will have no link with the PPTable

- ◆ Use icon to refresh the syntax
- ◆ Use icon to maximize the text zone

Insert a Local Machining Axis: General Process (2/2)

Outputs are computed in the current Machining Axis:

```
TLAXIS/ 0.000000, 0.000000, 1.000000
$$*CATIA0
$$ Manufacturing Program.1
$$    1.00000    0.00000    0.00000    54.13857
$$    0.00000    1.00000    0.00000   -33.03097
$$    0.00000    0.00000    1.00000    73.00000
SPINDL/  70.0000,RPM,CLW
GOTO/ -46.32305,   38.67889,  -25.00000
GOTO/ -33.82813,   38.67889,  -25.00000

...
TLAXIS/ 1.000000, 0.000000, 0.000000
$$  Start generation of: Change-Axis
$$*CATIA0
$$ Change-Axis
$$    0.00000    0.00000    1.00000    74.13857
$$    1.00000    0.00000    0.00000   -33.03097
$$    0.00000    1.00000    0.00000    0.00000
SPINDL/  70.0000,RPM,CLW
GOTO/  41.86405,   28.93750,  -25.00000
GOTO/  48.44451,   28.93750,  -25.00000
```
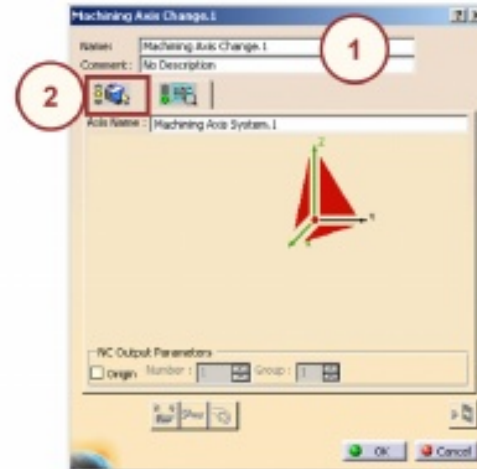
Manufacturing Program.1
Tool Change.1 T1 End Mill D 10
Pocketing.1 (Computed)
Change-Axis
Pocketing.2 (Computed)
Machining Process List.1
uctList
urcesList

Machining Axis System.1
Machining Axis System.2

First Machining Axis:
Defined on the Part
Operation

Second Machining Axis:
Defined through a
Machining Axis operation
(Change-Axis in the tree)

Insert a Post-Processor Instruction: General Process

1 Type the Name of the PP instruction and a comment line (Optional)

2 Type the Post-Processor Instructions to be generated or
Select PP Word and pre-defined syntaxes using the PP table access capability

3 Confirm operation creation

The Post-Processor Instructions will be generated in the APT following exactly the format that you have used to define them
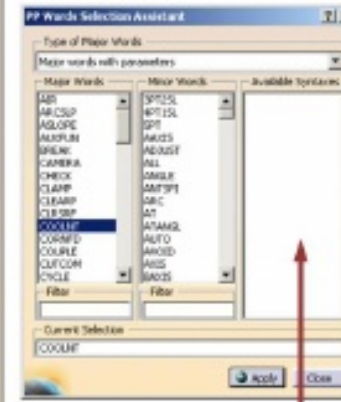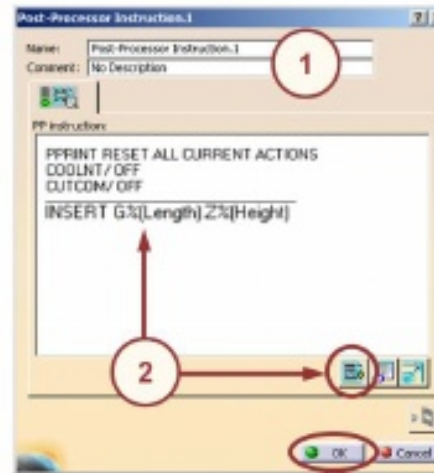
Add words or syntaxes in PP instruction window

The result is the following in the APT Source:
GOTO/ 41.86405, 43.00000, -25.00000
PPRINT End of generation of : Pocketing.2
PPRINT OPERATION NAME : Post-Processor Instruction.1
PPRINT Start generation of : Post-Processor Instruction.1
PPRINT RESET ALL CURRENT ACTIONS
COOLNT/ OFF
CUTCOM/ OFF

INSERT G80.5 Z75.8
PPRINT End of generation of : Post-Processor Instruction.1

Generate PP instructions with reference to Parameters
◆ **Using the "%" keyword**
◆ **Design Parameters**
◆ **NC Parameters**
◆ **User Parameters**
Can also be added in the PP table

You can merge several PP instructions and edit PP words inside a tool path.

How to Use Copy Instruction (1/2)

It allows to copy a sequence of operation including Tool Change and PP instruction.

1. Select in the tree, the last operation you want to include in the copy
2 and 3. Define Start point for the copy:
   - Create an index instruction before the first operation you want to copy or
   - Select an existing index instruction for the start
4. Select your options:
   - Number of copies
   - Type of transformation
   - Parameters of the transformation
5. Click OK

COPY Operator.1

Name: COPY Operator.1
Comment: No Description

Index Number: 1

Number of copies (N): 3

Transformation type: Translation

Translation type: Absolute coordinates

Distance along X: 0mm

Distance along Y: 0mm

Distance along Z: -140mm

OK   Cancel

Tree:

P.P.R.
ProcessList
  Process
    Part Operation.1
      Manufacturing Program.1
        INDEX.1 (Index # 1)
        Tool Change.1 T1 End Mill D 10
          Pocketing.1 (Computed)
          Pocketing.2 (Computed)
          COPY Operator.1 (Index # 1)

With this functionality there is no automatic tool change creation mechanism. That means if you want to have a tool change you must include it in the transformation (so put the index instruction before the tool change)

How to Use Copy Instruction (2/2)

Strategy Panel:

1. Define index number
2. Index management toolbar:
    - create an index instruction (to define the beginning of the copy)
    - Create an index/NoMore instruction
    - Define the beginning of the copy: select the index instruction in the tree
3. Define Number of copies you want
4. Define the type of transformation you want:
5. Define the geometry and the parameter necessary for the transformation

Translation
Rotation
Mirror
Axis to axis
Scale
Affinity
Matrix

Both Panels:

- Tool Path replay
- Define the name
- Define a comment

Post Processor Panel:

It allows to verify the generated syntax

This is the syntax that will appear in the APT file if I decide to generate APT file without resolving copy/Tracut syntaxes (option: Copy/Tracut Processing: Yes)

How to Use Tracut Instruction (1/2)

It allows to modify the trajectory of an operation by applying a transformation on it. (no duplication)

1. **Select in the tree, the level of insertion of the tracut (just before the operation you want to modify)**
2. **Define the endpoint of the tracut:**
   - **Create or**
   - **Select**
3. **Select your options:**
   - **Type of transformation**
   - **Parameters of the transformation**
4. **Click OK**

Without Tracut

With Tracut

P.P.R.
ProcessList
Process
Part Operation.1
Manufacturing Program.1
Tool Change.1 T1 End Mill D 10
Pocketing.1 (Computed) ①
TRACUT Operator.1
Pocketing.2 (Computed) ②
TRACUT/NOMORE.1

TRACUT Operator.1

Name: TRACUT Operator.1
Comment: No Description

Transformation type: Translation
Translation type: Absolute coordinates
Distance along X: 0mm
Distance along Y: 0mm
Distance along Z: -140mm

OK   Cancel

How to Use Tracut Instruction (2/2)

Strategy Panel:

1. **Index management toolbar:**
   Create an index/NoMore instruction

2. **Define the type of transformation you want**

   - Translation
   - Rotation
   - Mirror
   - Axis to axis
   - Scale
   - Affinity
   - Matrix
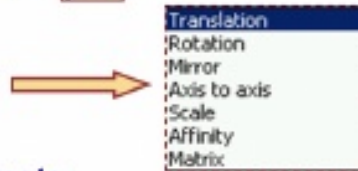
3. **Define the geometry and the parameter necessary for the transformation**

Both Panels:

- ◆ **Tool Path replay**
- ◆ **Define the name**
- ◆ **Define a comment**

Post Processor Panel:

It allows to verify the generated syntax

This is the syntax that will appear in the APT file if I decide to generate APT file without resolving copy/Tracut syntaxes. (option: Copy/Tracut Processing: Yes)
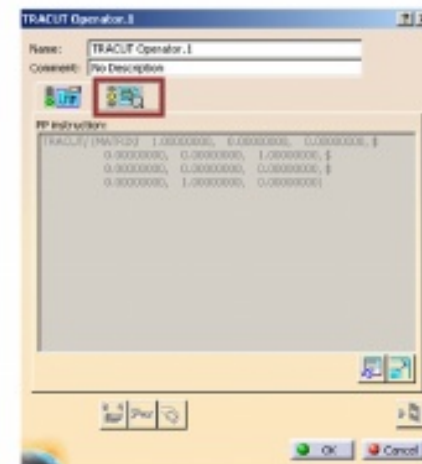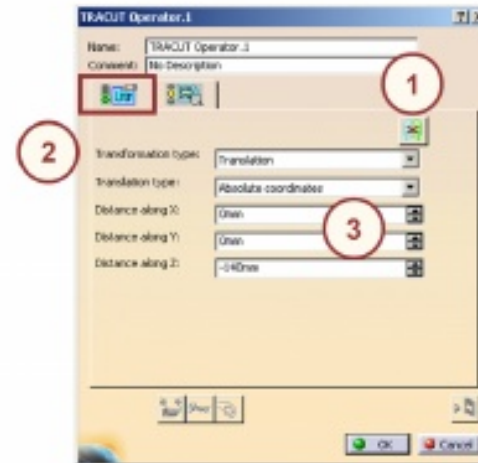
How to Use Copy Transformation Instruction (1/2)
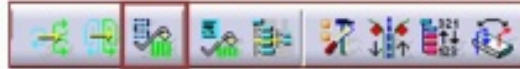
It allows to duplicate tool path by applying a transformation on it.
(available only for machining operation using the same tool)

1. **Select an operation to insert a new Copy-Transformation after it**
2. **Select a reference operation for the Copy-transformation**
3. **Name the operation and put a comment if necessary**
4. **Choose the transformation**
5. **Define transformation Parameters**
6. **Replay the tool path**

and

- Pocketing.2 (Computed)
- Facing.2 (Computed)
- Facing.3 (Computed)
- **1** Profile Contouring.1 (Computed)
- Copy-Transformation.1 (Computed)
  - Tool path
  - Tool path
  - Tool path
- Profile Contouring.2 (Computed)
- Profile Contouring.4 (Computed)
- Tool Change.3 T1 End Mill D 10-1
- Drilling Deep Hole.1 (Computed)

Contouring Opn

Copy transformation of the contouring Opn

Copy-Transformation.1

Name: Copy-Transformation.1
Comment: No Description   **3**

Name	Type
Profile Contouring.1	Profile Contouring

4

Number of copies (N): 3
Ordering: All operations N times
Transformation type: Translation
Translation type: Absolute coordinates   **5**
Distance along X: -20mm
Distance along Y: 0mm
Distance along Z: 0mm

6

OK   Cancel

- **No APT Tracut or Copy in APT output.**
- **It's the only transformation Operation which has its own tool path associated to it. So you have the possibility to edit and modify the trajectory.**

How to Use Copy Transformation Instruction (2/2)

Copy Transformation Instruction - Definition Panel:

1. **Management of the selected operation for copy, ability to**
   - ◆ **Add**
   - ◆ **Remove**
   - ◆ **Move**
   - ◆ **Sequence operations**

2. **Define the number of copies**

3. **Define the Ordering**

   All operations N times
   Each operation N times

4. **Define the type of transformation you want** ⟹

   Translation
   Rotation
   Mirror
   Axis to axis
   Scale
   Affinity
   Matrix

5. **Define the geometry and the parameters necessary for the transformation**

Copy-Transformation.1

| Name: | Copy-Transformation.1 |
| Comment: | No Description |

Name	Type
Pocketing.1	Pocketing
Pocketing.2	Pocketing

Number of copies (N):	1
Ordering:	All operations N times
Transformation type:	Translation
Translation type:	Absolute coordinates
Distance along X:	0mm
Distance along Y:	0mm
Distance along Z:	-140mm

OK   Cancel
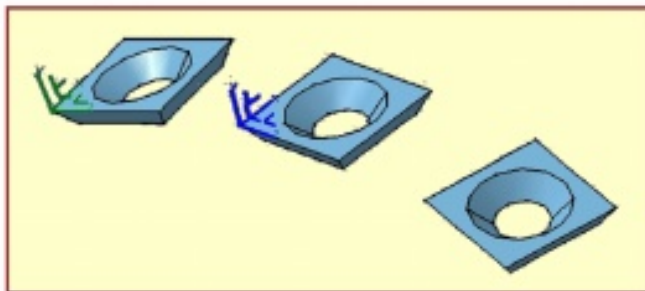
Define the name
Define a comment
Tool Path replay

Better Process Support: associative with initial operations, support cycle syntaxes and compensation.

All Instruction Details of Possible Transformations

The possible transformation types are as follows:
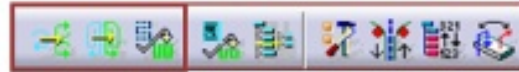
◆ **Translation:** choose the required translation type then specify the translation by:
   • either giving X, Y, Z components in the absolute or the current machining axis system or
   • clicking the Direction area to select a linear geometric element for the direction and giving a length.

◆ **Rotation:** select a linear geometric element as the axis of rotation then give a rotation angle. If a circular edge is selected, the normal axis of the circle is used.

◆ **Mirror:** select a planar geometric element as the axis of symmetry.

◆ **Axis to axis:** select a first machining axis system then select a second machining axis system. The first axis system will be transformed into the second axis system.

◆ **Affinity:** select a Machining Axis System and define 3 scale factors to be applied along each of its axes: x,y,z. The transformation matrix in the selected Machining Axis System will be: $\longrightarrow \begin{bmatrix} x & 0 & 0 & 0 \\ 0 & y & 0 & 0 \\ 0 & 0 & z & 0 \end{bmatrix}$

◆ **Scale:** select a planar surface or a point and a scale factor to be applied along the normal projection on the selected element.

◆ **Matrix:** This transformation will be defined by the matrix definition of the transformation in the absolute Axis System, the current Machining Axis System, or a selected Machining Axis System.

> In case of definition of the matrix in the absolute Axis System or in the current Machining Axis, the matrix of the transformation is stored in the model in the absolute Axis System (it is invariant in this Axis System).
> Choosing one or the other mode only changes the display of the coordinates of the matrix.
> Out of a current Machining Axis System context, the Absolute Axis System will be used to display the matrix.
>
> In case of definition of the matrix in a selected Machining Axis System, the matrix of the transformation is stored relatively to this selected Machining Axis System.

How to Use Copy Program
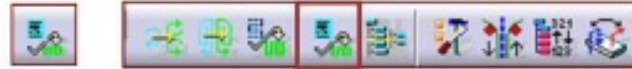
This functionality allows you the methodology for defining the process for machining identical or similar parts on one setup with only one process and a single tool list.

Copy Program: This functionality is the extension of the Copy Transformation function to Manufacturing Program.

1. **Select an insertion level (MP or MO) and click Copy program icon**

2. **Select the Manufacturing program to be transformed**
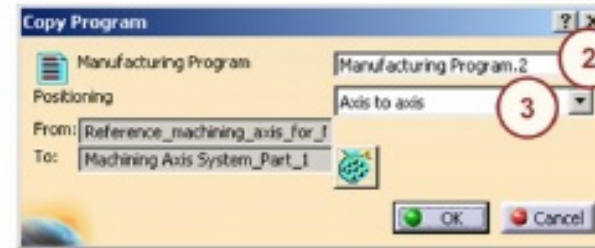
3. **Select the mode of the positioning**
   - ◆ **Axis to axis: It allows to define initial and final axis**
   - ◆ **Part to part: It allows to define initial and final part. The initial part is selected in the Product List of the tree. This transformation is equivalent to the Axis to Axis transformation.**

4. **Click OK.**

The Manufacturing program will be transformed with its tool changes, operations, machining axis changes and post processor Instructions.

Allows you to multi-select transformations

First part

Second part

Initial part

OR
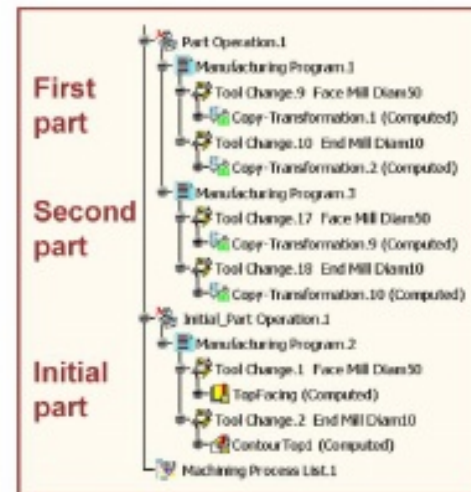
Transformed Programs

How to Use Merge Program
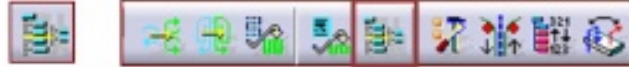
Merge Program: You can merge Manufacturing programs with optimization of the number of tool changes contained in the selected Machining Programs.

The order of the machining steps on the same part is maintained.

1. **Click Merge program icon**

2. **Click 'Add Programs' button and select the Manufacturing programs to be merged. The programs must be in the same Part Operation.**

3. **Click Preview button to see the list of tools in the resulting tool changes.**

4. **Click OK.**

All components of other selected manufacturing Programs will be added in the first selected program.
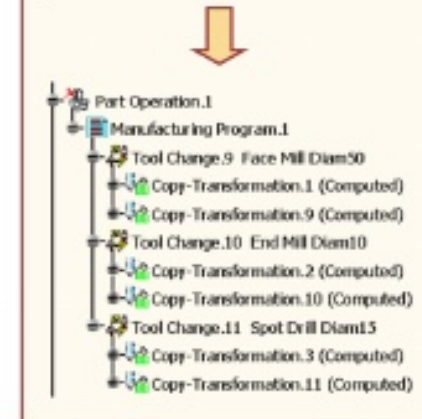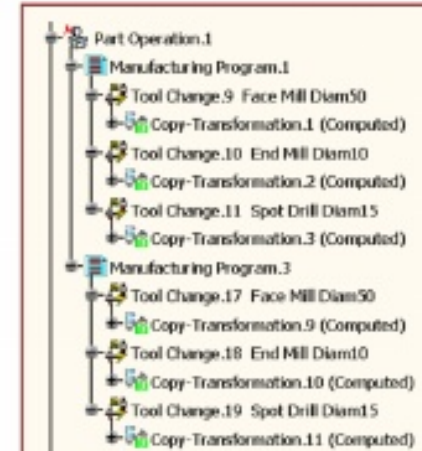
Merged Programs

Options for Tracut Operation in case of Symmetrical Part Machining
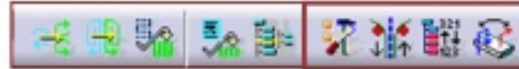
These options allow to manage symmetrical part machining using the inversion of machining direction.

Select the type of operation from the list, which you want to be impacted by the reverse machining conditions. The operations are grouped into three different categories - Prismatic, Surfacic and Multi-Axis operations.

Use this icon to reverse machining condition of the selected machining operation

Use this icon to reverse macro motions of the selected machining operation

Use this icon (Reorder Operations List) to reverse the order of one or more groups of operations

Opposite Hand Machining Options

Always apply 'Reverse Machining Conditions' command for the following operations:

Prismatic | Surfacic | Multi-Axis

[Select ALL for this tab]
- Multi-Axis Sweeping
- Multi-Axis Contour Driven
- Multi-Axis Tube Machining
- Multi-Axis Spiral Milling
- Multi-Axis Curve Machining
- Multi-Axis Flank Contouring
- Multi-Axis Helix Machining
- Multi-Pockets Flank Contouring

Close

Tool Change.1 T1 End Mill D 10
Pocketing.2 (Computed)
Tool path
TRACUT Operator.3
Pocketing.6-reversed (Computed)
TRACUT/NOMORE.2

Warning

One or more selected operations were locked. Do you want to unlock them and apply the Reverse Machining Conditions?

Yes   No

A keyword can be automatically added to an operation name when applying reverse machining condition with Tool/Options/Machining/Operation/Reverse keyword parameter. If any of the MOs are locked a message will appear as shown.

Auxiliary Operations Status

In the PPR Tree, for each Auxiliary Operation is associated a graphical icon which gives information to the user.

Tool Change Operation:

Operation created Automatically

Operation created Manually

Machine Rotation:

Operation created Automatically

Operation created Manually

All the Auxiliary Operations may have also the following status:

Operation Deactivated (done manually by the user)

Operation Not Complete (Some information is missing: geometry,etc)

Operation Not Updated (Tool path must be replayed to update the operation)

Output Generation

You will learn how to Generate Manufacturing Program Outputs.

- **General Process to Generate Output Files**
- **General Process to Generate NC code Output Files**
- **How to Generate HTML Documentation**
- **More Details of Generating NC code**
- **More Details about Batch Queue Manager**

General Process to Generate Output Files

Generate Outputs

At the end of the NC programming you will generates outputs:

- NC code
- Shop floor Documentation

NC code:

You can generate APT, CLFile, « G » or « ISO » code in three way:

- Interactively
- In Batch (you block your CATIA session
- In batch queue (deferred)

Shopfloor:
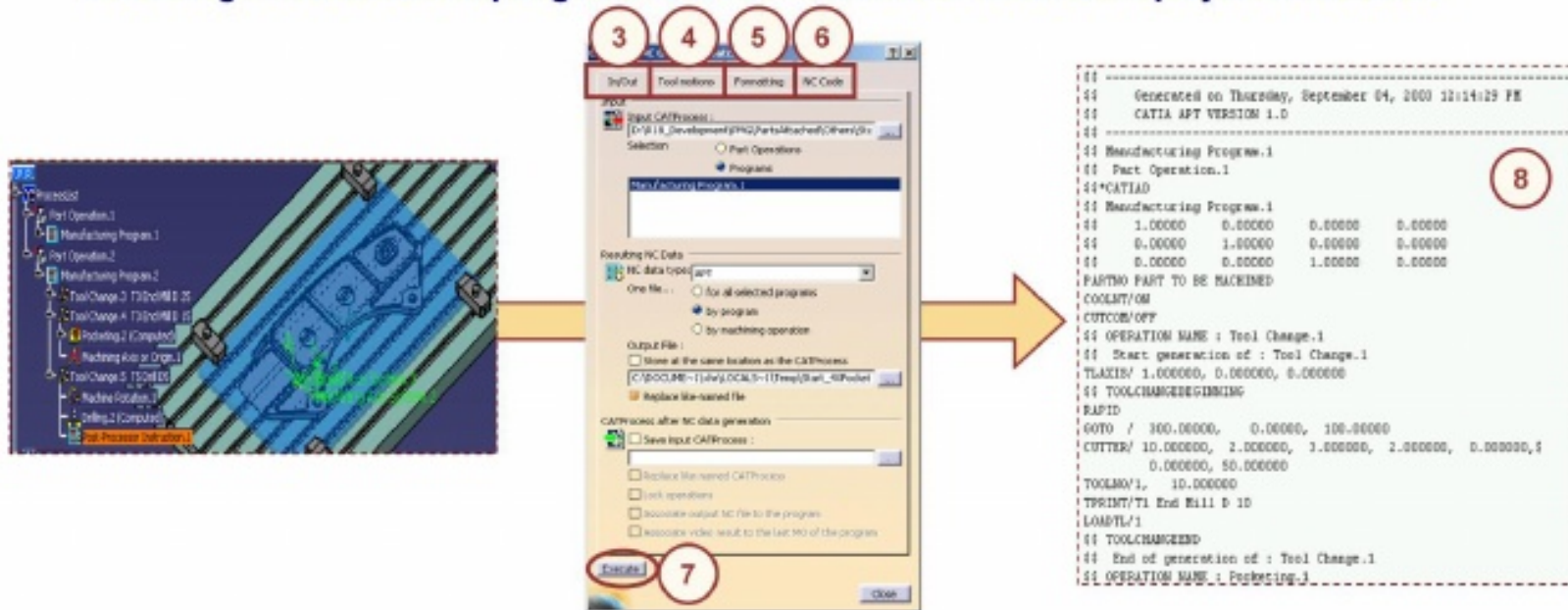
You can generate tool list + process list.

The functionality is a VB macro that you can customize.

General Process to Generate- NC Code Output Files

It allows you to generate APT, CLF, NC Code or CGR file.

1. Save the CATProcess before generating the APT Source Code (batch mode only)
2. Click on [ ] or [ ] to generate NC Code
3. Select the In/Out Tab Page to specify the Input and Output of the computation
4. Select Tool Motions Tab page to generate specific syntax
5. Select the Formatting Tab Page to specify some point coordinate format and comment statement
6. Select NC code tab page to define the post processor you want to use
7. Execute
8. A log file will be generated which contains the warning/Error message entries. A message indicating whether the Output generation is successful or failed is displayed to the user.

How to Generate HTML Documentation

1 **Click** 📄 **to generate the HTML Documentation**

2 **Specify in the Process Documentation dialog box the following information:**
- ◆ **Documentation script (CATScript document)**
- ◆ **Process type (only Process in this version)**
- ◆ **Folder where the documentation will be generated**
- ◆ **Name of the file**
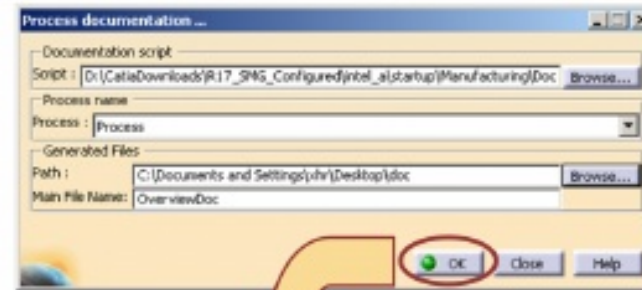
3 **Click OK to generate your documentation**

Documentation for Process: *ManufacturingProcess*

Generation Date : 6/14/00 at 11:48:16 AM

Setup Number: 1 , Name :MyFirstPartOperation
Setup Comment: NoDescription
Setup Tool Catalog Name: ToolsSample01

Program Number: 1 , Name: TrainingProgram
Program Comment : NoDescription
Program description

Activity number	Activity name	Tool number	Activity type	Spindle speed	Feedrate
1	Tool Change 1	1	ToolChange	x	x
2	Pocketing 1	1	Pocketing	500	200
3	Tool Change 4	2	ToolChange	x	x
4	Drilling 1	2	Drilling	500	200

Some samples and helps are delivered under intel_a/startup/Manufacturing/Documentation of your CATIA V5 installation.
You can create your own Script in VB SCRIPT and then run it using macro standard execution.

Generating NC Code: More Details (1/5)

Generate NC Output: In/Out tab page

1- Select the program to process:
- ♦ Select the document
- ♦ Select the Part Operation or the machining Program

2- Select the resulting NC Data:
- ♦ The NC Data Type (APT, CLFile, NC Code or Video result in CATProduct)
- ♦ Split or not the output file by program or operations
- ♦ The Output File name
- ♦ Store the file in the CATProcess directory instead of the one define in the settings
- ♦ tag « replace file » to crash the old NC output version file

3- If needed, you can choose to save automatically the document (CATProcess) after processing.
- ♦ Select the Save document check box and specify
- ♦ Where you want to save it, using the « ... » button
- ♦ Replace the old CATProcess
- ♦ Lock the operation of the CATProcess automatically
- ♦ Associate document: create a link between the CATProcess and the code generated.
- ♦ Associate the generated CATProduct with the last machining operation of each program or the last program.

Batch Mode: You need to write the CATProcess before generating the APT Source, CLF, CGR file, but during the computation, your CATIA V5 session is available.

Interactive Mode: You don't need to write the CATProcess to generate the APT Source Code, but your CATIA V5 session will be blocked for the duration of the computation
The APT Source Code can be also generated by right-clicking on the Manufacturing Program.
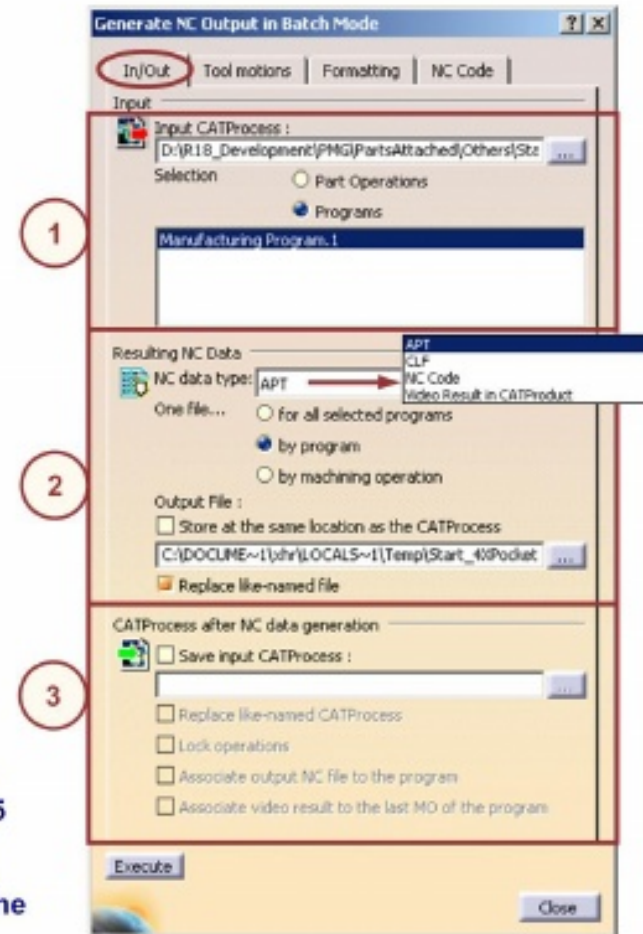
Generating NC Code: More Details (2/5)

Generate NC Output: Tool motions tab page
Tool motions parameters:
Home Point strategy:
You can choose to include Home Point information in the NC data output by means of this option. In this case GOTO or FROM information defined on the part operation's machine is used.

Include GOTO for tool change:
For each tool change, generate GOTO instruction (to the tool change point define in the PO)

Output Cycle syntaxes:
The PP word syntax specified in the PP word table will be output for axial machining operations instead of GOTO statements.

Remove GOTO before cycles:
For axial machining operations using SYNTAX output mode (CYCLE), you can now choose whether or not to output GOTO statements corresponding to Jump and Clearance motions (points that were added by the clearance approach distance or by the jump distance)

Process COPY and TRACUT operations:
Copy and/or Tracut instructions will be processed. In this case there will be no Copy or Tracut statements remaining in the generated APT source

Remove double point after PP Commands:
First point after PP command or user syntax is not kept if the previous one is a coincident point

Remove aligned points:
You will keep only the first and the last

From machine: The information is automatically retrieved on the Machine-Tool associated to the current Part Operation.
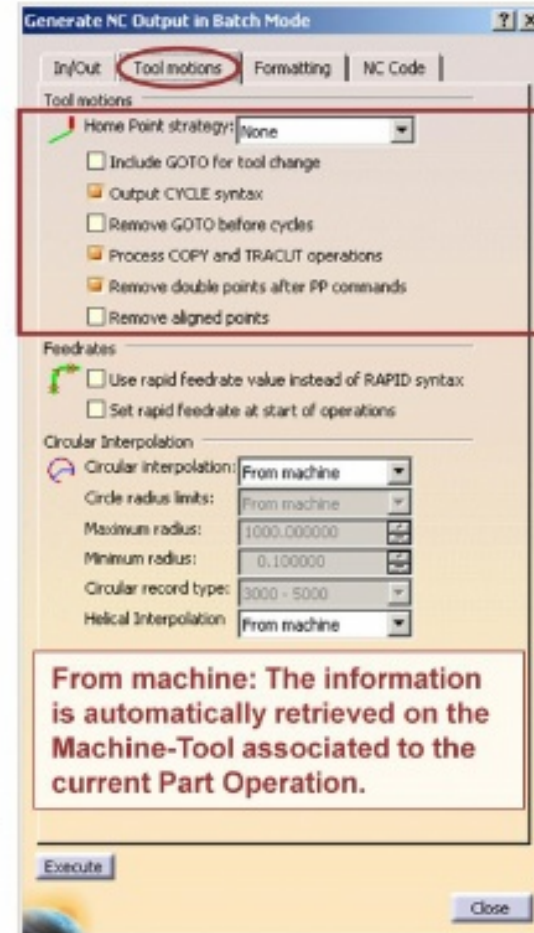
Generating NC Code: More Details (3/5)

Generate NC Output: Tool motions tab page

Feedrates parameters:

Use rapid feedrate value instead of RAPID syntax:
Rapid motions will be preceded by a FEEDRATE syntax whose value is the Rapid feedrate specified on the machine.

Set rapid feedrate at start of operation:
A RAPID statement will be included at the start of each operation. However, if a Clearance macro is defined on an operation, the macro definition will be taken into account.

CIRCULAR INTERPOLATION:
Specifies the type of circles to be processed if circular interpolation is required:
- **From machine:** uses the values specified by the part operation's machine
- **None:** circular interpolation is not required
- **Z-axis circles:** only circles whose axis is parallel to the z-axis of the machining axis system are processed
- **Any axis circles:** all circle types are processed

Circle radius limits:
Specifies how circles are to be processed for circular interpolation:
- **From machine:** the values specified by the part operation's machine are used
- **Value:** user-defined values are used for minimum and maximum radius constraints

Maximum radius, Minimum Radius, Circular record type:
Specifies the type of record to be generated on the clfile if circular interpolation is requested

Helical Interpolation
Specifies the type of helix to be processed if helical interpolation is required.

Click **Execute** to request computation of the APT Source file
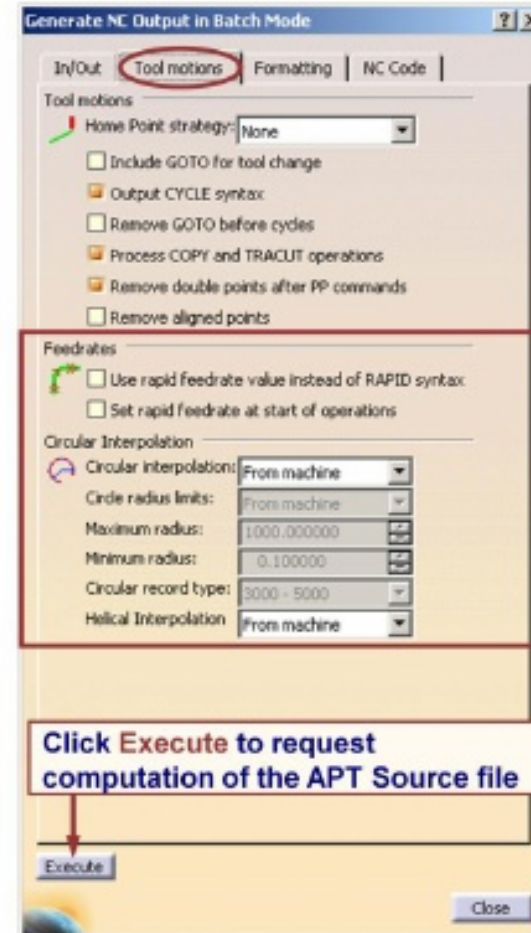
Generating NC Code: More Details (4/5)

Generate Output: Formatting tab page

A. Statements:

Tool motion statement:

Defines the format describing tool motion statements on the NC data output:

- **From machine: the output format defined the part operation's machine is used.**
- **Point: tool point coordinates (x,y,z) are output.**
  A TLAXIS statement is given at the start of the generated APT source.
  A fixed-axis clfile record 9000 is given at the start of the generated clfile.
- **Axis: tool point coordinates and tool axis components (x,y,z,i,j,k) are output.**
  A MULTAX statement is given at the start of the generated APT source.
  A MULTAX clfile record 9000 is given at the start of the generated clfile.

General information, Part operation, Machining operation name:

Defines how information will be generated:

- **None: not generated**
- **PPRINT: generated with the PPRINT word**
- **$$: generated as a comment (not available for clfile).**

B. Format For Points coordinates (X,Y,Z):

Allows you to define other formats for NC data statements allowing better accuracy for large parts

- **Number of digits**
- **Digits after decimal**

C. Format for axial components (I,J,K):

- **Number of digits**
- **Digits after decimal**
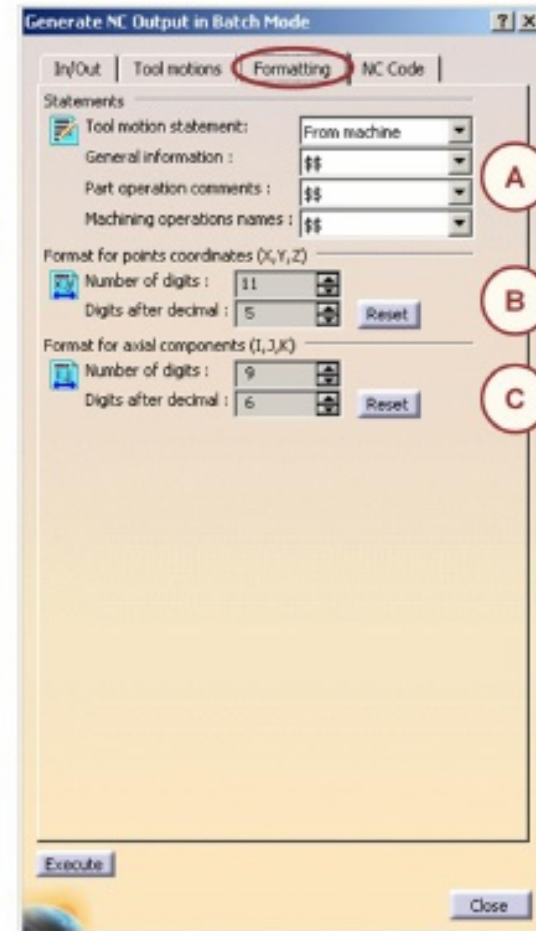
Generating NC Code: More Details (5/5)

Generate Output: NC Code tab page

1 Use the same procedure as to generate APT Source Code but specify NC Code type for the Output format in the In / Out Tab Page

2 Select the NC Code Tab Page to specify the name of the Post-Processor to use for ISO NC Code generation

Click Execute to request computation of the ISO NC Code

1 NC Output Interactively **2**

In/Out | Tool motions | Formatting | NC Code

Cenit Post-processor file

SINUMERIK_840D_3X.pp
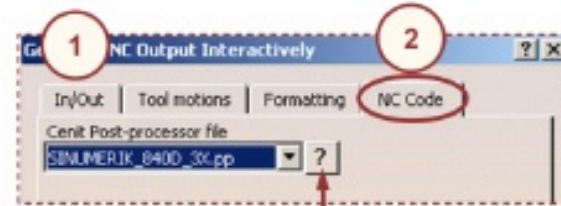
Click ? to access the documentation of the selected Post-Processor.
This online documentation includes the Post-Processor definition and NC data samples

Select the type of Post Processor solution using « Tools > Options > Machining > » Output tab. If the output option is set to None, you will not be able to generate NC code.

Sample Post Processor parameter files are delivered with the product in the folder:

◆ for CENIT: \Startup\Manufacturing\PPPar

◆ for IMS: \Startup\Manufacturing\IMSPar (The IMSPar folder must be accessible in Read-Write mode)

◆ for ICAM: \Startup\Manufacturing\ICAMPar

To execute your own PP you must copy it into these folders.

Machining time value is indicated in the log file

```
Information: activity Spot Drilling.1 has been updated.
Information: ** Machining and total time  : 0.088 , 0.114322 min. **
Information: ** activity Spot Drilling.1 : END . **
Information: ** activity Manufacturing Program.1 : END . **
Generation of NC code file : End.
```

More Details About Batch Queue Manager

Batch Queue Manager:

- Ability to generate output file (ISO, APT, CLFILE)
- Each job can have a different output file
- Each defined job can have a different output file type
- (APT, NC Code)
- CATMFG Options Panel can be accessed
- Ability to generate one file per MO

Define a New job

Select the MP or the PO in the tree

NC Batch Management

Process	Selection	Description	Status	Comment
D:\R18_Development\PMG\Pa...	Manufacturing Program.1	Compute		Job to generate APT file

• Deferred: the computation will start at the designated time

• Immediate: the computation will start as soon as you click the Activate button.

Batch mode ○ Deferred ● Immediate

Job management functions (New, edit, move, delete)

For delay option, Program start computation

Always save your program modifications before generating the NC code. For best results, you must first verify the operations of your program by means of a replay or simulation. There must be no operations to be updated or in an undefined state.

Advanced Topics

Following advanced topics are covered:

- **Import and Modify Tool Path**
- **Aerospace Structure Part- (AdvEX:00)**
- **Import V4 NC Mill and NC Lathe Set**
- **Aerospace Structure Part- (AdvEX:01)**
- **Machining Processes**
- **Aerospace Structure Part- (AdvEX:02)**
- **Manage Resources**
- **Aerospace Structure Part- (AdvEX:03)**
- **Aerospace Structure Part- (AdvEX:04)**
- **PP Word Table Customization**
- **Aerospace Structure Part- (AdvEX:05)**
- **Design Change Management**
- **Aerospace Structure Part- (AdvEX:06)**

Import and Modify Tool Path

Following topics are covered:

- General Process to Import APT / CLF / NC Code Files
- General Process to Modify a Tool Path
- Tool Path Management
- Tool Path Management: More Details

General Process to Import APT / CLF / NC Code Files

This functionality is available from NC Manufacturing Review Workbench.

It allows to import APT in CATIA and then you can,

- ◆ **Replay them**
- ◆ **Simulate them (if you associate a Stock to the Part Operation)**
- ◆ **Modify with tool path editor**

1. **Select a Manufacturing Program**
2. **Click "NC File Import" icon**
3. **Select NC Data type (APT or CLFile or NC Code)**
4. **Click Input File**
5. **Choose your PP (activate before your PP Supplier in the menu Tools/Option)**
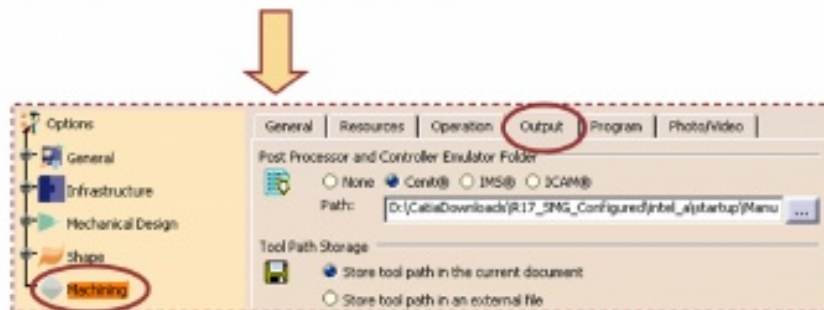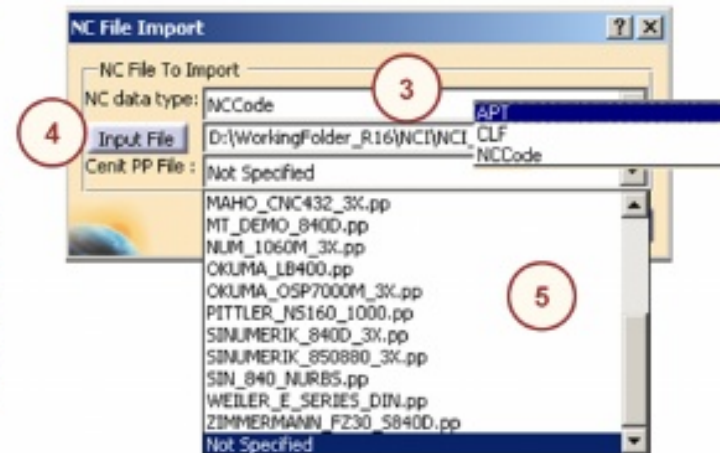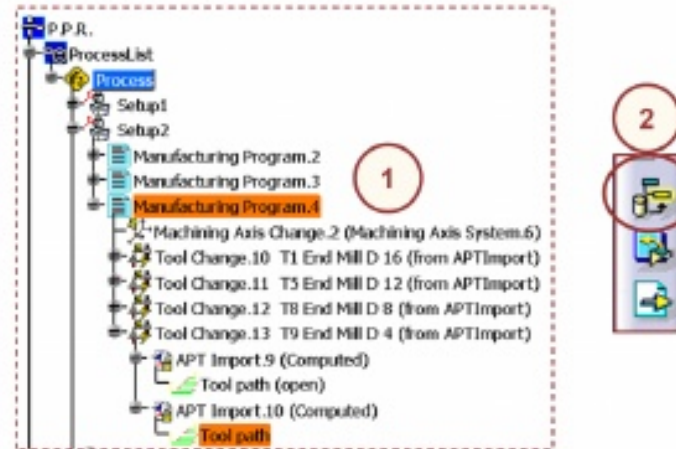
General Process to Modify a Tool Path

🔹 **You have the possibility to modify a Tool Path**

  ◆ **After APT import**

  ◆ **After Machining Operation computation**

🔹 **How to Access to the Tool Path Editor**

  ◆ **Select the Machining Operation (MO)**

  ◆ **Check that the tool path is unpacked**

    ▪ **If it is not: MB3 on the MO and select unpack**

  ◆ **Lock the Operation**

    ▪ **MB3 on the MO and select lock**

  ◆ **Modify the tool path**

    ▪ **Select the tool path in the PPR Tree**

    ▪ **MB3 on the tool path and select the type of modification or click icon in 'Tool Path Management' toolbar**
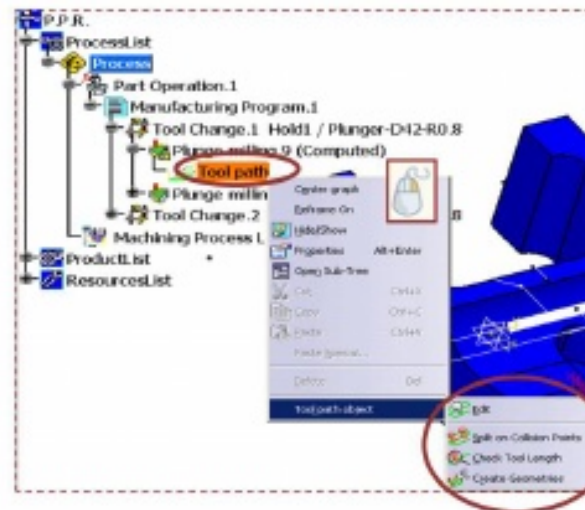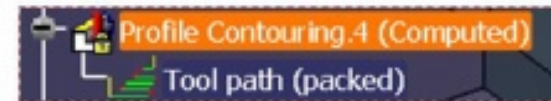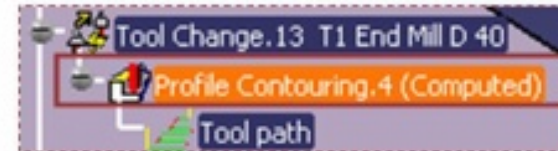
Tool Path Management

Tool path editor functions can be accessed using a toolbar "Tool path management".
This toolbar contains following commands:

Edit Tool Path:
All the functions with which the tool path can be edited, are combined in 'Edit Tool Path' toolbar are given below:

Point Modification: Point on the tool path can be moved or removed by selecting those points.

Area Modification: Area of the tool path can be modified after selection of that area.

PP Word Modification: It allows you to select previous & next PP Word, delete or edit PP Word.

Translation:

Rotation:

Mirror:

Transformations can be applied to a tool path.

Reverse: Tool path can be reversed but not displayed. Approach and Retract points are exchanged.

Connection: Tool path can be connected.

Approach and Retract Modification: Approaches and Retracts can be added or removed from tool path.

Points Display mode: Allows to hide the points on tool path display for Point modification, Area modification, Rotation.

The functions which work on the tool path, but do not intend to modify it are:

Split on Collision Points:
Longer Tool path splits according to specified or longer tool.

Create Geometries:
Using tool path, geometry can be previewed and/or created.

Check Tool Length:
A tool path is checked to identify all the points where the tool or the tool/holder collides with the part.

Tool Path Management: More Details

You will learn in detail about Tool Path Management.

- Point Modification
- Area Modification
- Translating a Tool Path
- Rotating a Tool Path
- Mirror
- Splitting on a Collision Point
- Connecting Tool Paths
- Changing Approach and Retract
- Checking Tool Length

Numerical Control Infrastructure

STUDENT GUIDE

Student Notes:

Point Modification

The functionality allows you to move or delete a selected point on a tool path.

- Multi selection of point
- Selection by sweep
- Selection between two points
- Selection by polygonal trap

- Reverse selection
- Reset selection
- Cuts the current points
- Confirm the modification
- Inserting a point

Point Modification dialog:
Selection / Action
☐ To represent circles
X: 80mm
Y: 0mm
Z: 59.991mm
OK   Cancel

Once the points are selected, you can move them:

- Pull the Distance arrow to the place you want the point to be in the viewer. The distance between the original position and the current position of the points is displayed as you move the arrow or

- Type the coordinates where they must be in the spin boxes. Just as above, an arrow is displayed as well as the distance from the original position of the points or

- Double-click the word Distance and type the distance in the box.

Use the contextual menu on Distance to select the translation direction.

Distance arrow

Distance = 11.645mm

Distance dialog:
Distance : 11.645mm
OK   Cancel

Along X axis
Along Y axis
Along Z axis
Along tool axis
Along last polyline
Along next polyline
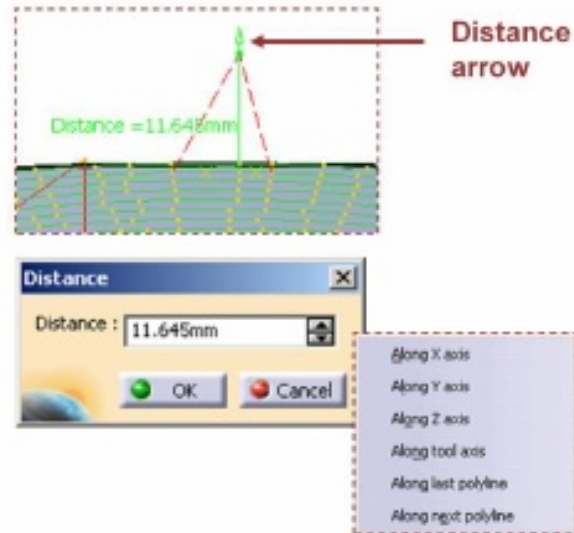
Area Modification (1/2)

You can edit the area of a tool path. Area can be selected using several editing functionalities. Area modification is used to correct the tool path which is discontinuous or irregular.

Tool Path Editor

Selection between two points

Selection between two points

Selection by one point

Selection by contour

Selection by polyline

selected points

Selection by one point

selected point

'Cancel' button in Point/Area modification and Approach & Retract Modification allows canceling all the modifications done inside the dialog box.
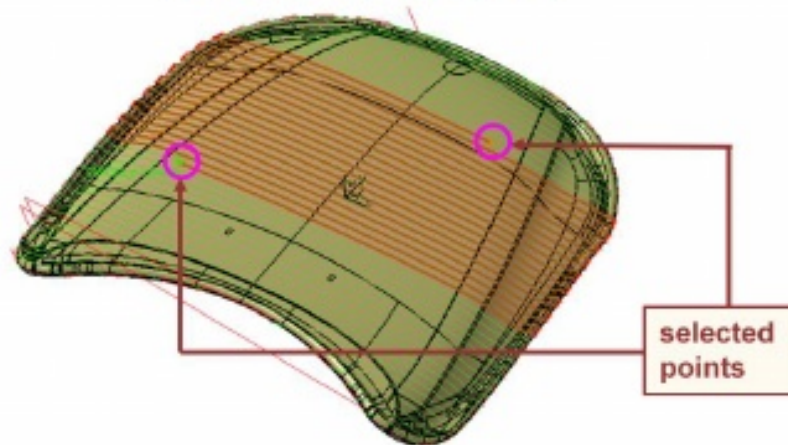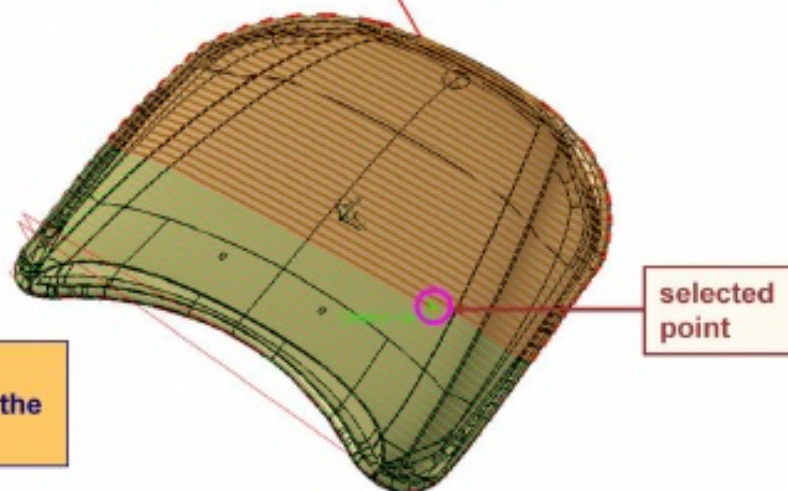
Area Modification (2/2)

Select collision points

Reverse selection

Cut the current points

Validate the modification

Area selection option

Modify the feedrate

You can change the feedrate of a partial tool path to approach, retract, machining or local.

Before cutting an area of the tool path, you can choose to copy this area in the specification tree. Copy transformation check box need to be selected and click OK.

Selection Option

Selection Mode
- Before/In selected geometry
- After/Out selected geometry

Cut destination
- Copy transformation

OK   Cancel

Selection by contour

closed contour

Selection by polyline

polyline
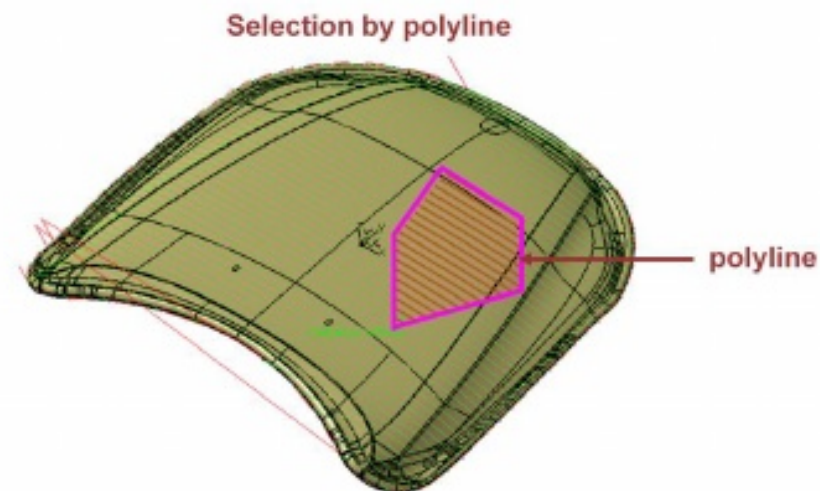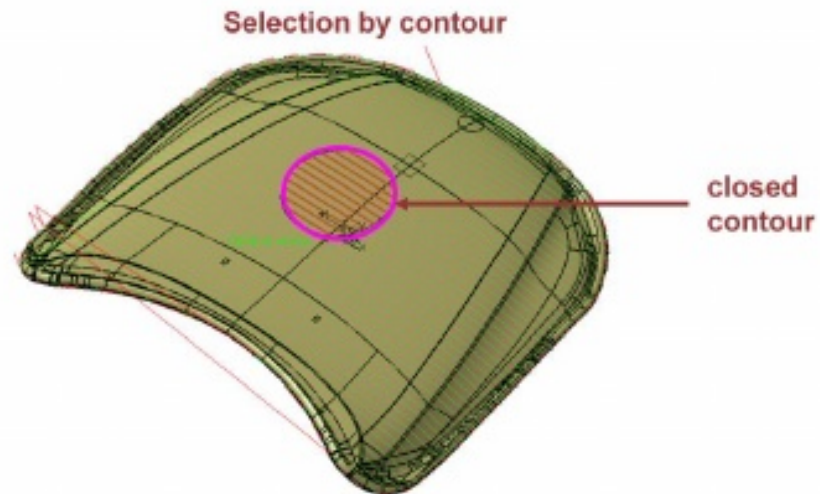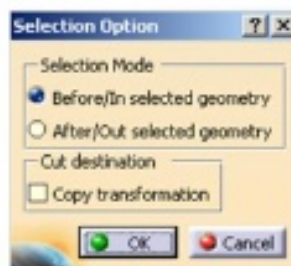
Translating a Tool Path

You can translate the tool path using this functionality. The distance by which the tool path to be translated can be entered through double-clicking on distance value or by dragging the distance arrow in required direction.

1. Click Translation button in 'Edit Tool Path' icon. The tool path is displayed on the part.

2. You can translate the tool path by dragging from approach or retract. The contextual menu over the word 'Distance' allows you to select the axis for translation of the tool path among:
   - The X axis,
   - The Y axis,
   - The Z axis, or
   - The tool axis.

3. And then pulling the tool path, you can also double-click Distance and specify a value in the distance dialog box that is displayed.

4. Double-click anywhere in viewer to translate the tool path and exit the action.

translated tool path
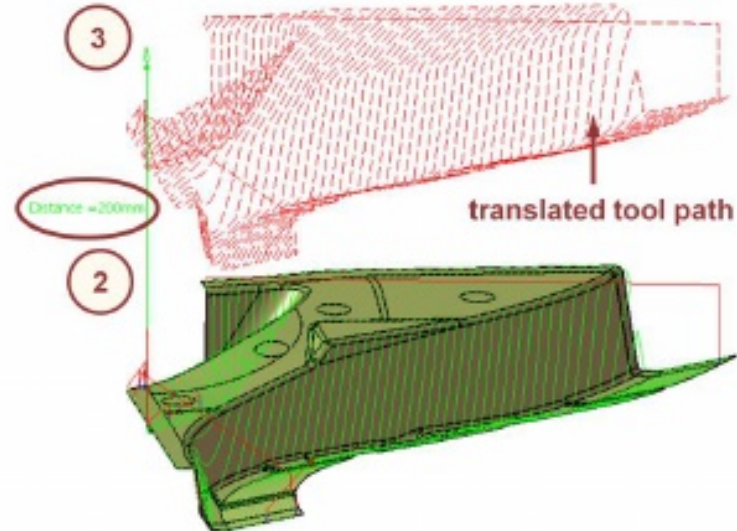
Rotating a Tool Path

The functionality allows you to rotate the tool path by any angle with reference to a point, a edge, a plane or a face.

1. Click **Rotation** button in 'Edit Tool Path' icon. The tool path is displayed on the part.

2. You can define the rotation you want with respect to:
   - A point: this defines the origin for the rotation,
   - An edge this defines the rotation axis,
   - A plane: the normal to the plane defines the rotation axis or
   - A face: the normal to the face defines the rotation axis.

   As you move the mouse over the tool path, the elements that can be used for the rotation are highlighted in red. By default the rotation is effected around the tool axis.

3. Change the angle by double-clicking on the word 'Angle' in the viewer (you can also drag the direction arrow in the viewer). A dialog box is displayed. Type the number of degrees you want to rotate the tool path by.

4. Double-click anywhere in viewer to rotate the tool path and exit the action.

rotated tool path
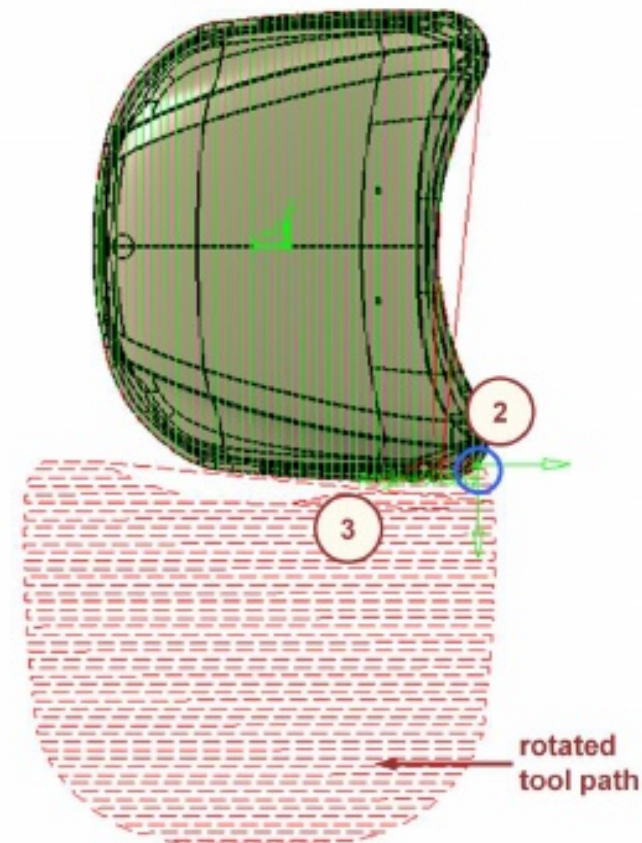
Mirroring a Tool Path

The functionality allows you to mirror the tool path
With respect to a plane or a face.

1. Click **Mirror** button in 'Edit Tool Path' icon.

2. Select a plane or a face as a mirror plane.
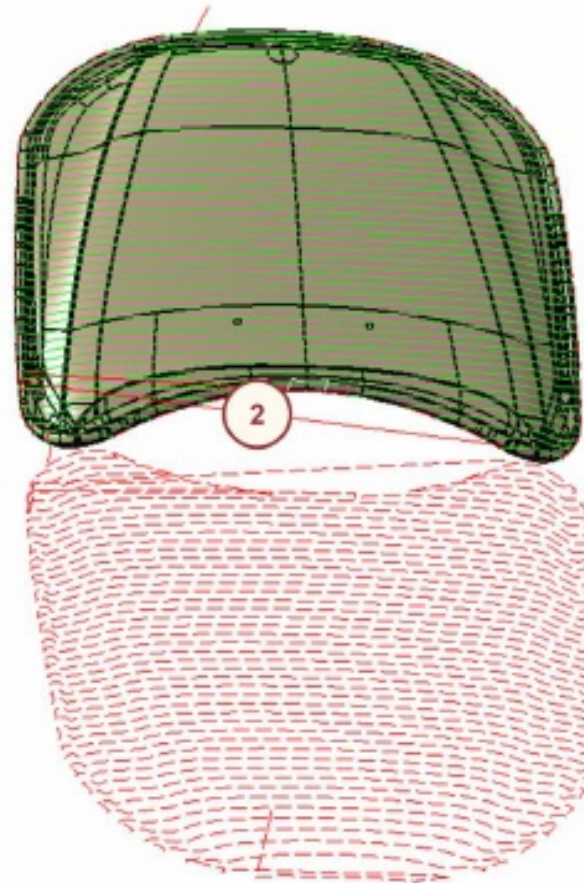
3. Double-click anywhere in the viewer to mirror the tool path and exit the action.

Splitting on a Collision Point

Split of tool path on collision points is required when the tool length is a constraint. The tool path can be split according to the specified tool or a longer tool.

1. Once you have set the parameters, click **Apply**.
   The points in collision appear in red.

2. Now select a longer tool in the **New tool list**.
   This tool length could be computed using

3. Confirm the creation. A **Copy-Transformation** containing the points in collision is created in the specification tree with a tool path that is computed with the new tool.

4. Now close both tool paths using **connection** or **change approach/retract** option.

Collision points

Manufacturing Program.1
  Tool Change.7 T7 Ball nose D 10
    Sweeping.2 (Computed)
      Tool path (open)
  Tool Change.8 T3 Ball nose D 10
    Copy-Transformation.2 (Computed)
      Tool path (open)

174
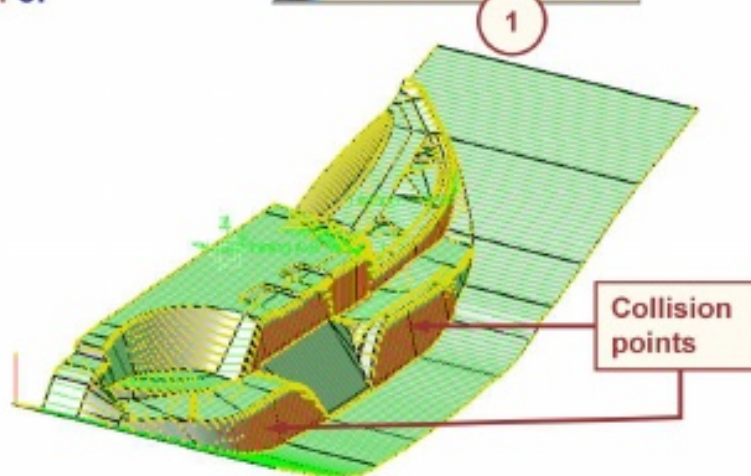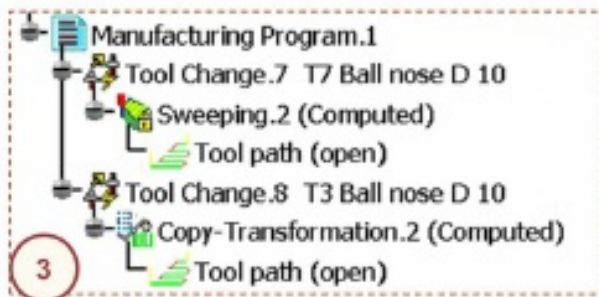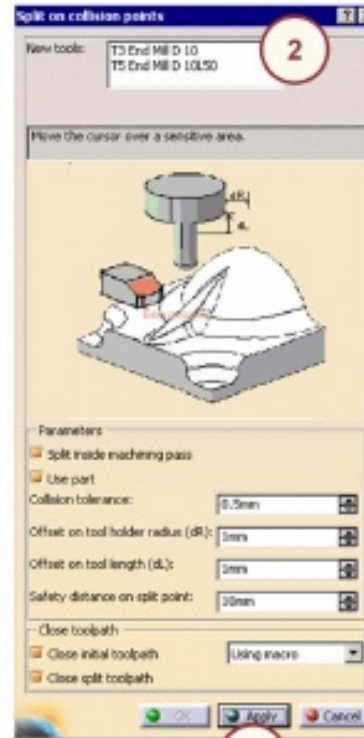
Connecting Tool Paths

Tool paths which are split for the modification need to be reconnected. This functionality helps you to connect tool paths to maintain the continuity. Hence gaps in the tool path are removed and gouging of tool in material is avoided.

Multi-selection of point

Selection by sweep

Selection between two points

Selection by polygonal trap

Reverse selection

Reset selection

Straight connection

Plane connection

Safety plane connection
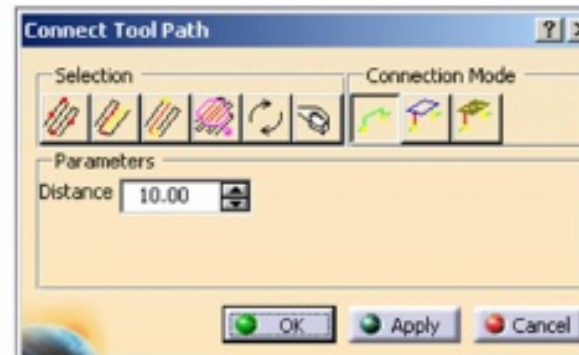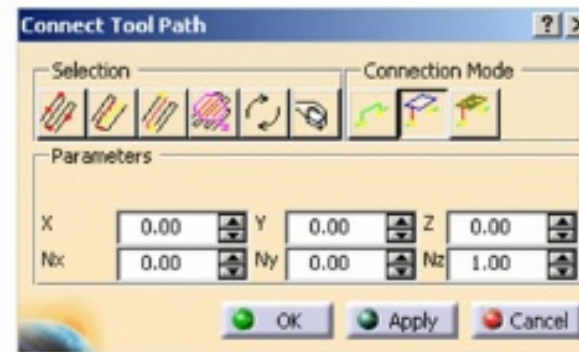
The safety plane must be selected either in the current operation or on the part operation.

Straight Connection

Plane Connection

Changing Approach and Retract

You will learn how to add, remove or modify approaches & retracts in a tool path.

Modification of approaches and retracts

Delete

Filter
- Approach
- Retract
- Linking passes
- Between paths

Remove from whole tool path
Remove from area inside polygon

Add / Modify

Selection

Apply

Approach | Retract

Mode: Back ?

Back motion

Distance: 10mm ?

Height: 6mm ?
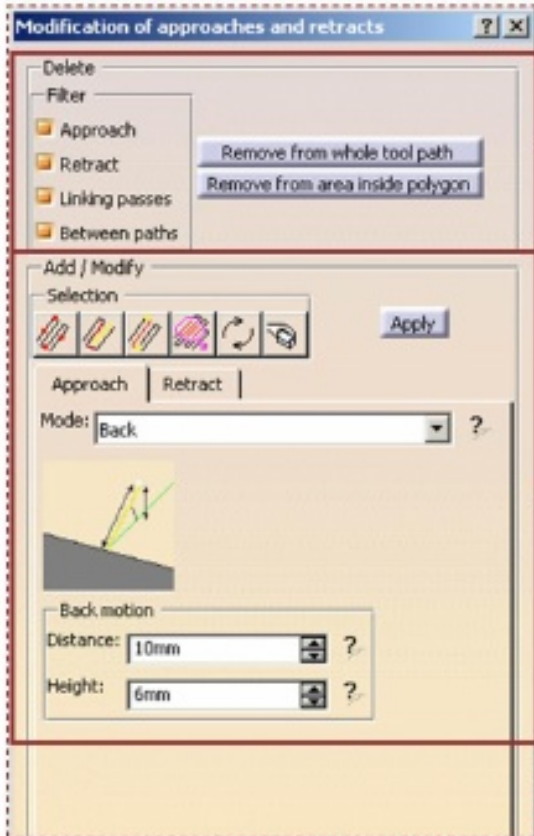
You can Delete:
- ◆ **Approach,**
- ◆ **Retract,**
- ◆ **Linking passes,**
- ◆ **Between paths.**

from the whole tool path or from a polygon that you draw on the tool path.

You can Add/Modify:
- ◆ **Approach,**
- ◆ **Retract**

1. **Choose the Approach or the Retract tab.**
2. **Select the type of motion you want to use and modify the settings if necessary.**
3. **Press Apply. A message is displayed:**

Whole toolpath selected

(i) Apply to the whole tool path?

Yes   No
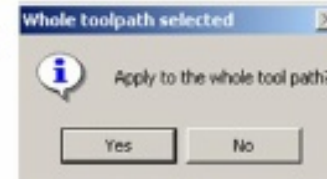
If you answer Yes, you will add an approach or a retract motion to the whole path. If you answer No, use the Selection bar to define an area to apply the approach or retract motion.

4. **When you are satisfied with the results press OK. If not, continue to make changes to the approach and retract tabs till you get satisfied.**

Checking Tool Length

This functionality explains how to check a tool path to identify all the points where the tool holder collides with the part. If you consider the tool alone, only the cutting length of the tool is taken into account. If you consider the tool with its tool holder, the tool gage and the cutting length are taken into account.

1. Once you have set the parameters, click **Apply**. The points in collision appear in red.

2. A small dialog box is displayed that gives the number of collision points on this tool path, the minimum tool length that is required in order to avoid having collision points and the coordinates of the current point.

Tool length results

Collision

☑ Display tool

Input data
Tool length (Lc) = 10mm
Offset on tool length = 1mm

Results
Number of collision points = 784
Minimum tool length (Lc) = 55.108mm

Current point
X=35.565mm Y=-127.013mm Z=18.16mm
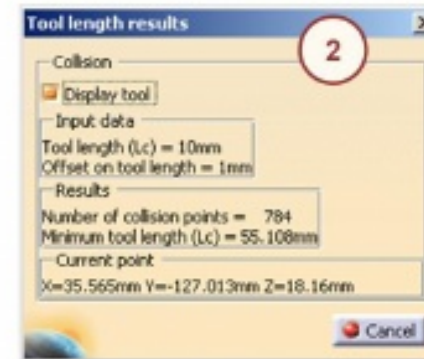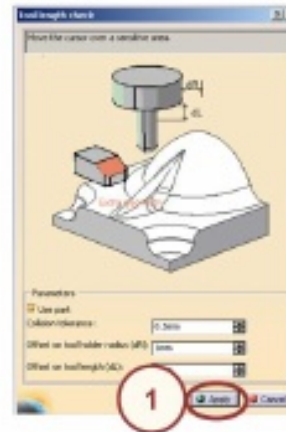
Mouse click gives the coordinates of the point
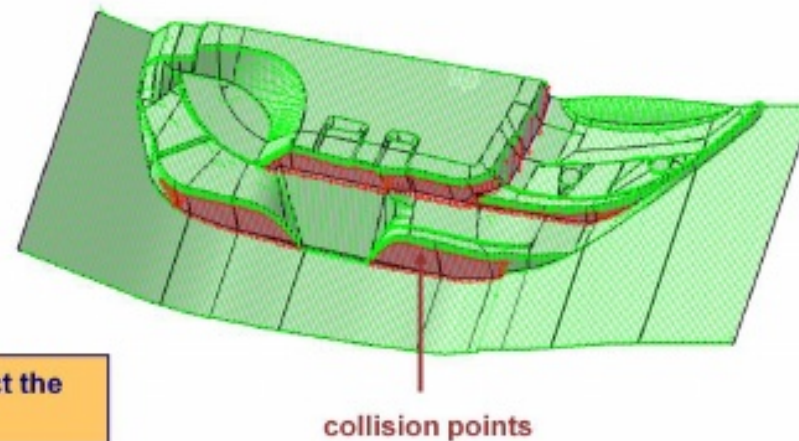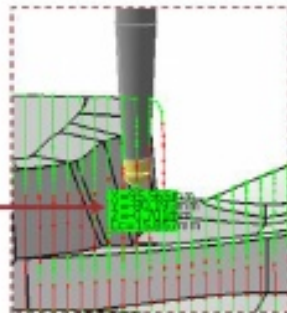
By this visual check, you can decide whether to select the proper length tool or to modify the tool path itself.

collision points

Import V4 NC Mill and NC Lathe Set

In this lesson, you will see how to import NCMILL and NCLATHE Set.

- Introduction to CATIA V5 Import CATProcess
- Import V4 Model (NCMILL/NCLATHE Set)

This Chapter is only for those people who are interested by Manufacturing Program Review and Migration V4 NCMILL Sets or V4 NCLATHE set -> CATIA V5.

If your are not interested go directly to the Next Lesson.

Introduction to CATIA V5 Import CATProcess

NC Review is the V5 NC Manufacturing Infrastructure then you can open, edit and replay all V5 CATPocess.

Browse V5 NC Operations with the same editor NC Products

Read V5 CATProcess

Access to the Replay Package if the Operation is Computed

Access to the Tool Path Editor If the operation is Computed

Generate NC Code in the required format

Produce Shop-floor Documentation
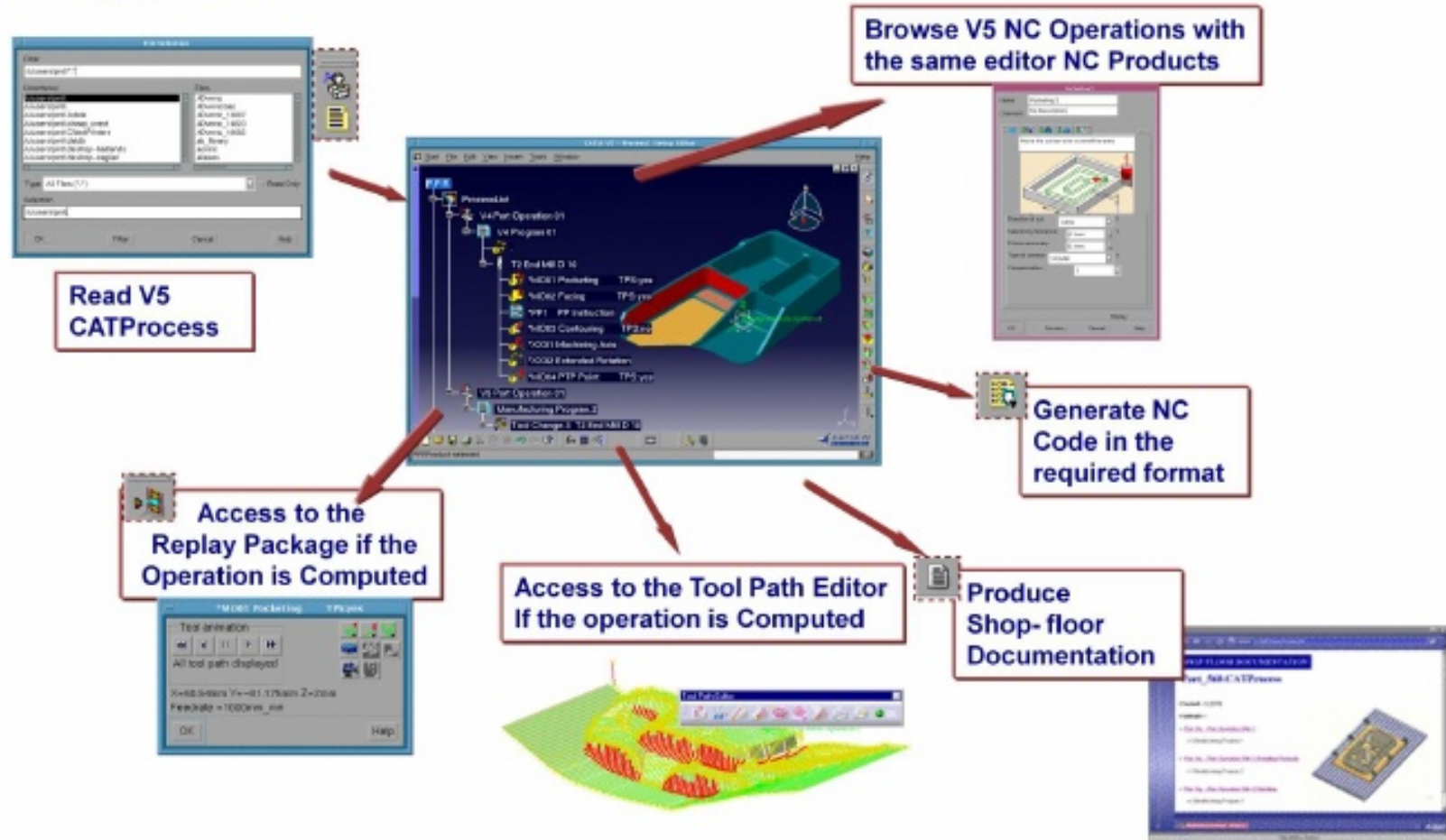
How to Import V4 Model- NCMill / NCLathe Sets (1/5)

Prepare your data:

1. **Insert your model in an empty CATProduct (use « insert existing component »)**
2. **Save your CATProduct**
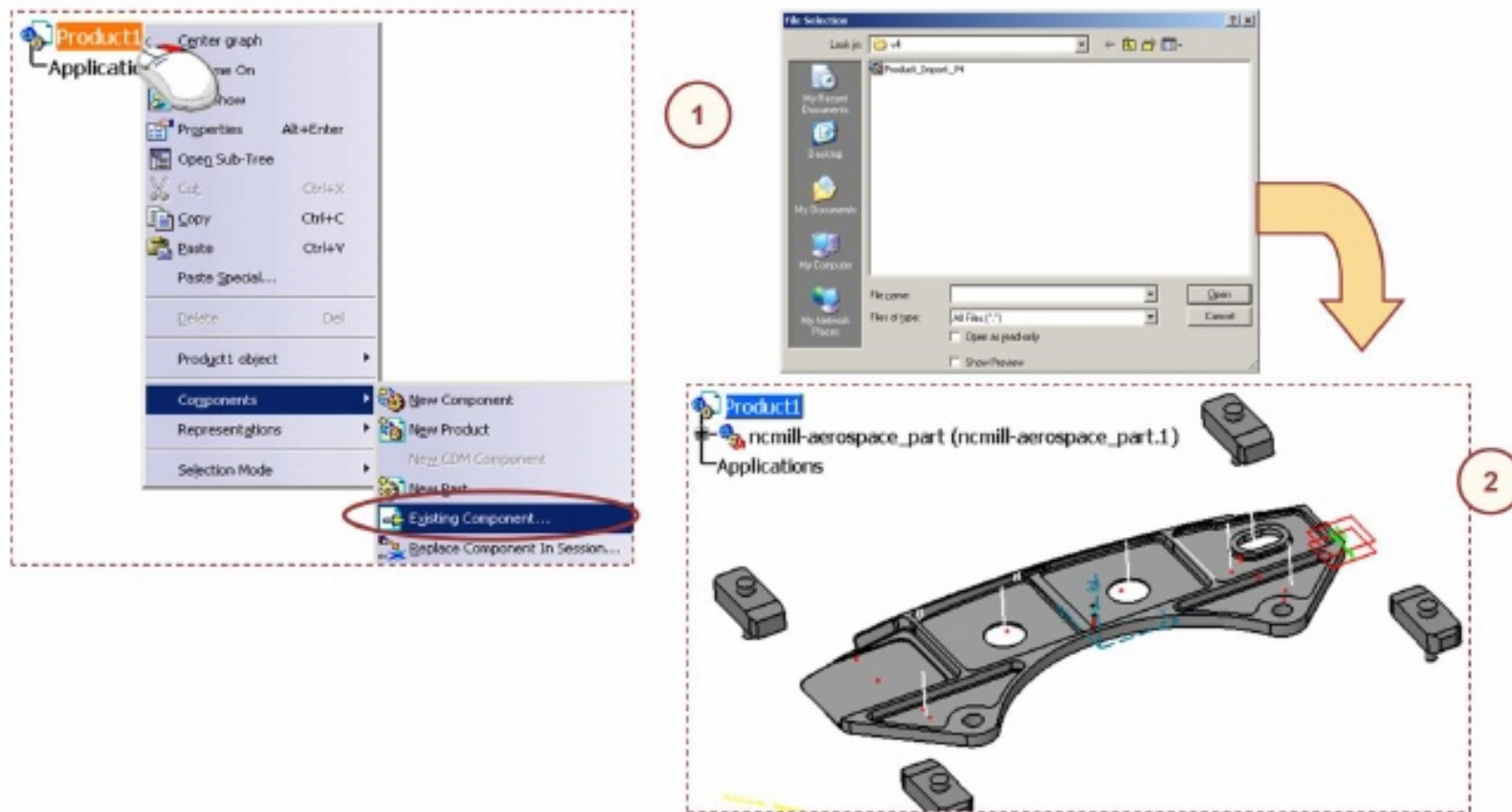
How to Import V4 Model- NC Mill / NC Lathe Sets (2/5)

Import your V4 NCMill and NCLathe SET:

1. **Access to NC Review Workbench.**
2. **Click "Read Manufacturing data from V4 model" icon.**
3. **Select your CATProduct which contain V4 model and the Sets.**
   If you have more than one V4 model in the CATProduct, select the V4 model you want to open.
4. **Select the set you want to import.**
5. **Click OK.**
6. **A new Part operation appears in the process tree with the associated Machining Program and operations.**

In the same time you can see a geometrical representation of the part.
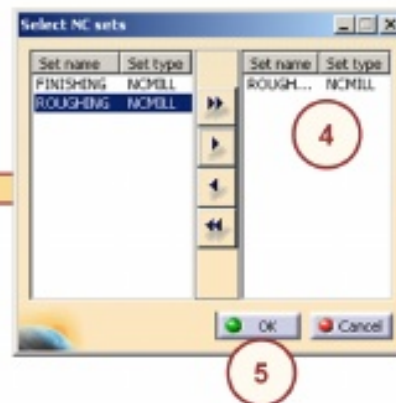
How to Import V4 Model- NC Mill / NC Lathe Sets (3/5)

Verify the Machining Operation Parameters and Geometry:

1. **Double-click the Operation**

2. **Check your V4 parameters**
   - ◆ **Strategy**
   - ◆ **Geometry**
   - ◆ **Tools**
   - ◆ **Feeds & Speeds**
   - ◆ **Macros**

1
- Part Operation.1
  - ROUGHING
    - V4 Program.1
      - 2 - TOOL CHANGE T1 End Mill D 16
      - 3 - STARTING POINT
      - 4 - ROUGHING

4 - ROUGHING

Name:	4 - ROUGHING
Comment:	No Description

2

Flank type :	Cylindrical
End type :	Filleted
Tool height :	35mm
Body Radius :	8mm
Corner Radius :	2mm
Cutter radius :	8mm

Cutter statement:
CUTTER/16,2,6,2,0,0,35
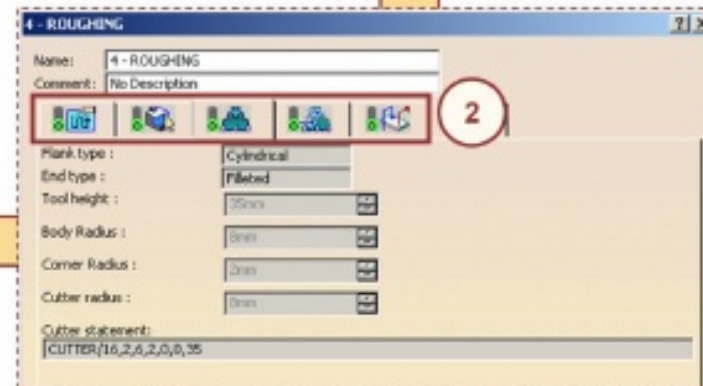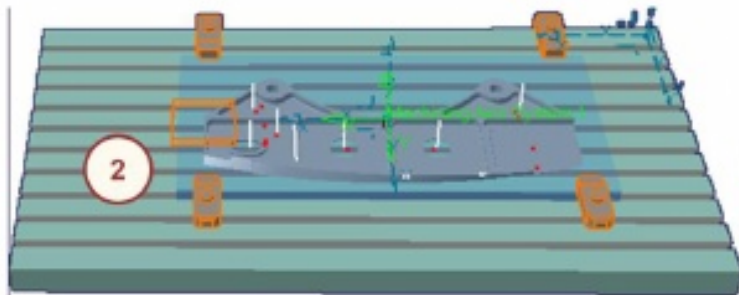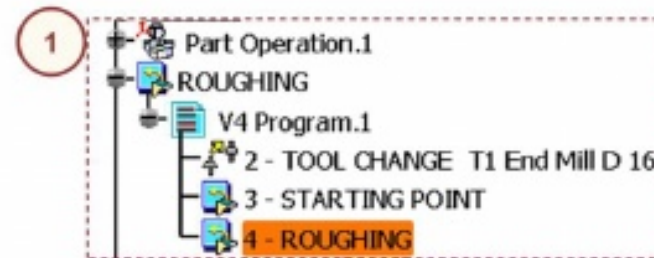
How to Import V4 Model- NC Mill / NC Lathe Sets (4/5)

Associate Output NC File to a Program

1. Click "Associate Output NC File to a Program" icon
2. Select your V4 Manufacturing Program
3. Select the AptSource file and click Open
4. A « Computed» comment appear on the tree near each operation

Replay the program to see the tool paths

Part Operation.1
ROUGHING
 V4 Program.1
  2 - TOOL CHANGE T1 End Mill D 16
  3 - STARTING POINT
  4 - ROUGHING

Part Operation.1
ROUGHING
 V4 Program.1
  2 - TOOL CHANGE T1 End Mill D 16
  3 - STARTING POINT (Computed)
  4 - ROUGHING (Computed)
   Tool path
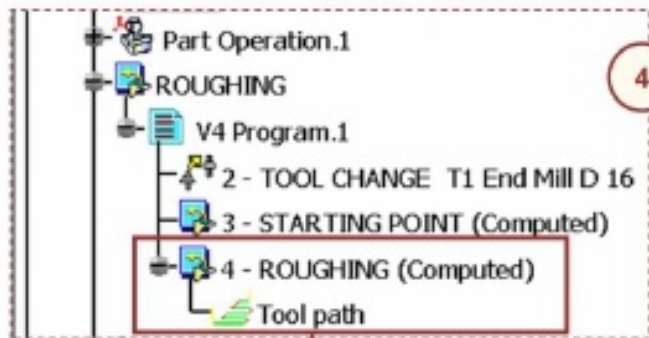
You can edit the tool path of V4 NCMILL operation using Tool Path Editor.
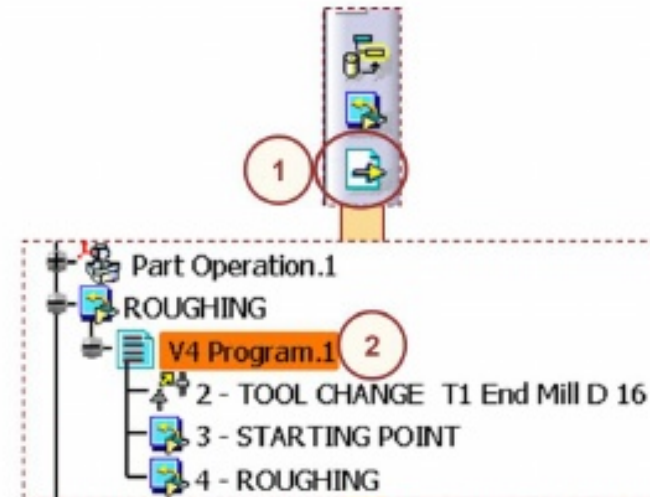
How to Import V4 Model- NC Mill / NC Lathe Sets (5/5)

Important Point:

- If you want to simulate (video mode) your tool path or add V5 Machining Operation; you need to have V5 Geometry to define stock, fixture in the PO and other element in the MO.

- In this case you must follow this Methodology:

    In your CATProduct:

    - Insert a new CATPart
    - In this new CATPart Copy / Paste as Result your V4 Model or only some elements of your model
    - In this way you can insert different CATPart in your CATProduct to store separately Part, Stock, Fixture
    - Now you can come back in the CATProcess and define the PO

Machining Processes

In this lesson, you will learn how to Create, Save and Reuse Machining Processes.

- Different Machining Processes
- Creating a Machining Process
- Storage in Catalogs and Reuse Of Machining Processes

Rudiments: Different Machining Processes

In a Machining Process you can Store:

- **A Predefined operation**
- **A Set of predefined operations (machining process)**

where you have set:

- ◆ **Parameters (fixes or according design or tool parameters)**
- ◆ **Macros**
- ◆ **Tool query (in hard or according design parameters)**
- ◆ **Formula**
- ◆ **Check**

This Machining Process is stored in a catalog.

You can instantiate it in your current session from the catalog. In this case, you have 2 possibilities

- ◆ **Select geometry or predefined set of geometry during the instantiation (axial processes, 3axes processes) in the model or in the tree or in the manufacturing view**
- ◆ **No Select geometry, so you need to affect it after manually for all the operations**

Creating a Machining Process (1/4)

- **Open a new CATProcess (1)**
  - ◆ **You need to create different Machining Operations without geometry**

In View Menu, active the Machining Process Tool Bar (2)
- **Click Machining Process View icon**

- **Create a New machining process** and rename it (3)

- **Create your sequence of MO (4)**
  - ◆ **Select the 1st Machining operation & click ok**
  - ◆ **Select the 2nd machining operation & click ok**

- **Define the parameters / Macros (5)**
  - ◆ **Double-click the operation**

- **Define the tool query thinks formula (6)**
  - ◆ **Double-click Tool query**

- **Define formula to set NC parameters according to design parameters (7)**
  - ◆ **MB3 on the operation "Edit Formula"**

- **Define Check formula to set condition (8)**
  - ◆ **MB3 on the operation "Edit Check"**

Knowledgeware Integration

Creating a Machining Process (2/4)

The Tool Query:

1. **Select the Tool catalog where the system must look in**
2. **Select the type of tool you want for the MO**
3. **Select the parameter to compare for the query**
4. **Select the operator for comparison**
5. **Key the value you want (number, name) or**
6. **Select the design or manufacturing parameter**
   - **Select the dictionary**
   - **Select the type of the feature**
   - **Double-click the parameter attribute**
7. **Use the button:**
   - **Up arrow to validate the selection**
   - **Down arrow to modify a formula**
   - **Cross to delete all the formula**
   - **Gum to delete the selected formula**
8. **Click OK**
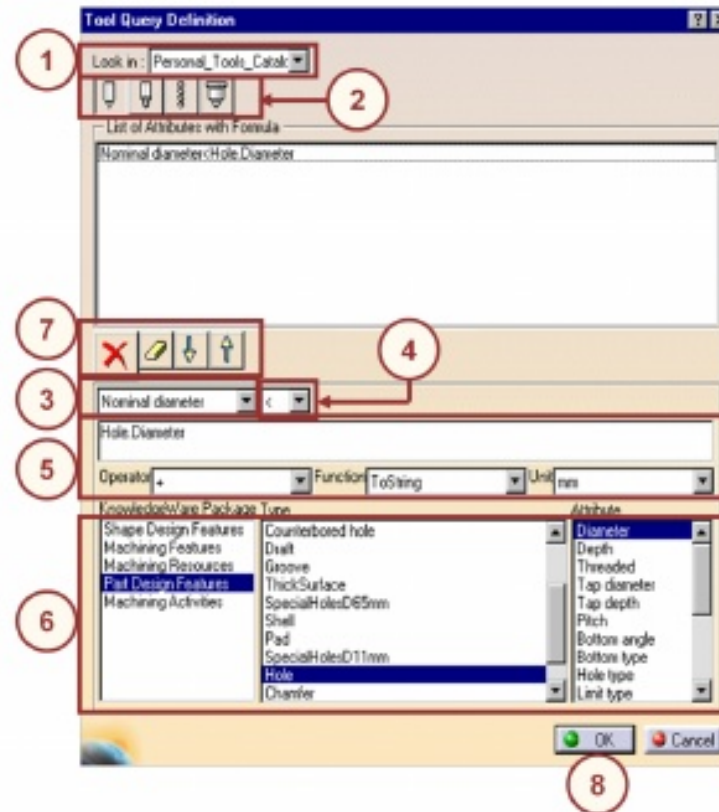
Creating a Machining Process (3/4)

Edit formula:

(set parameters according to design parameters or other Manufacturing parameters)

1. **Select the parameter to compare for the query**
2. **Select the operator for comparison**
3. **Key the value you want (number, name)**
4. **Select the design or manufacturing parameter**
   - ◆ **Select the dictionary**
   - ◆ **Select the type of the feature**
   - ◆ **Double-click the parameter attribute**
5. **Use the button:**
   - ◆ **Up arrow to validate the selection**
   - ◆ **Down arrow to modify a formula**
   - ◆ **Cross to delete all the formula**
   - ◆ **Gum to delete the selected formula**
6. **Click OK**

Using for UDF MP
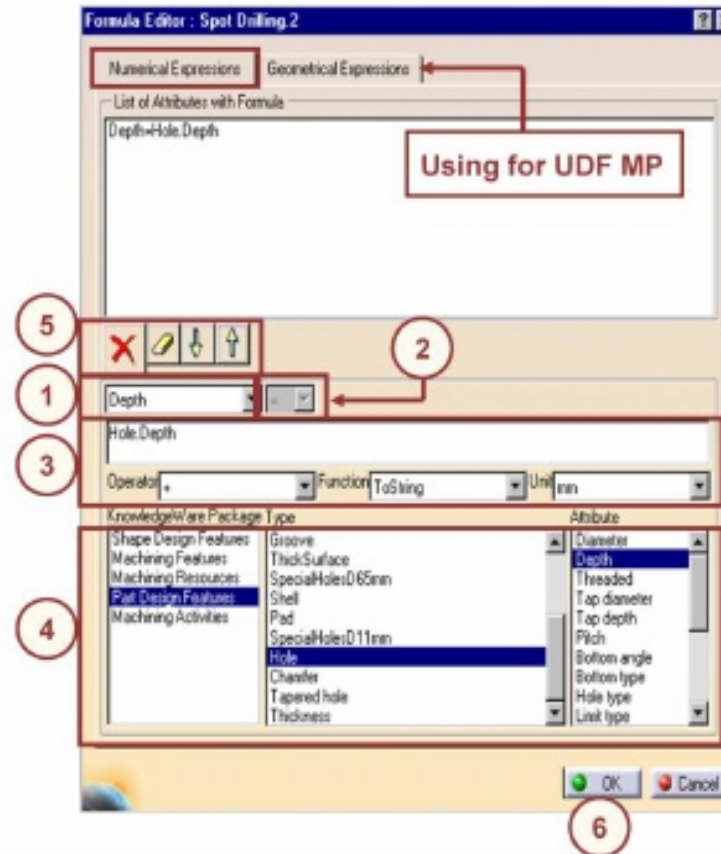
Creating a Machining Process (4/4)

Edit Check:

This formula allows to activate the operation only if a condition on a parameter is true.

e.g. the tapping operation will be imported only if the design hole is threaded.

1. **Select the design or manufacturing parameter**
   - **Select the dictionary**
   - **Select the type of the feature**
   - **Double-click the parameter attribute**
2. **Key the value you want (number, name)**
3. **Use the button:**
   - **Up arrow to validate the selection**
   - **Down arrow to modify a formula**
   - **Cross to delete all the formula**
   - **Gum to delete the selected formula**
4. **Click OK**
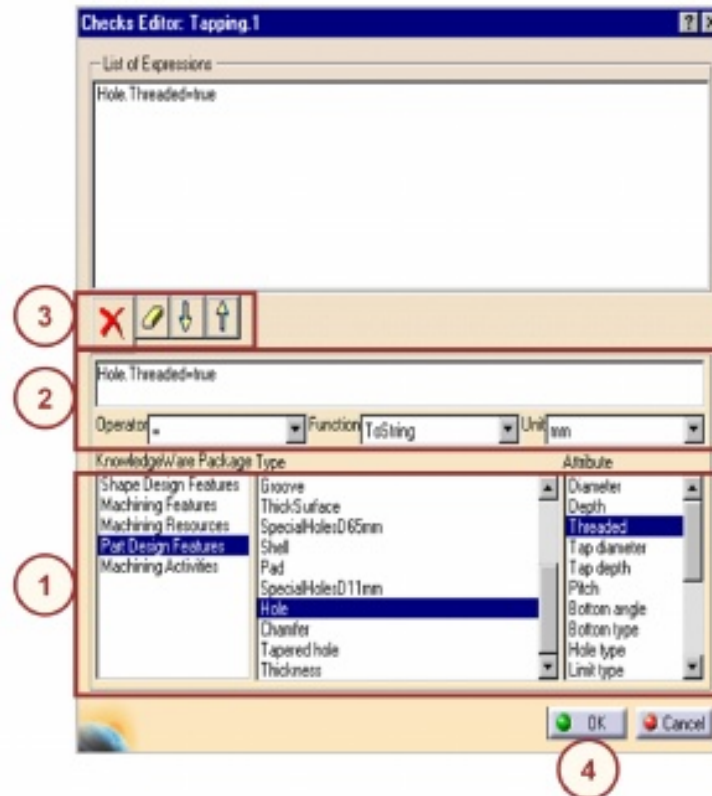
Rudiments: Storage in Catalogs and Reuse of Machining Processes (1/3)

- Use Catalog editor to store the Machining Process. To do so:
    - ◆ Save the CATProcess containing the Machining Processes (don't close it)
    - ◆ Create a new Catalog with Catalog Editor
    - ◆ Create your catalog organization (directory and family)------(A)
    - ◆ Add a Component and press "Select external chapter" -------(B)
    - ◆ Split your windows and select you machining process in the manufacturing view-------(C)
    - ◆ Validate: the Machining Process is saving in this catalog
    - ◆ Save the catalog
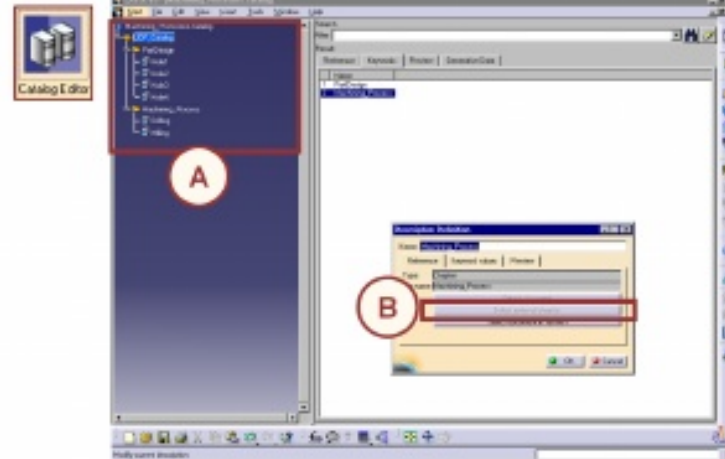
- Use « send to » functionality as for macro (easier):
    - ◆ In the machining process view, MB3 on the Machining process
    - ◆ Select save in catalog
    - ◆ Select the catalog or create a new one
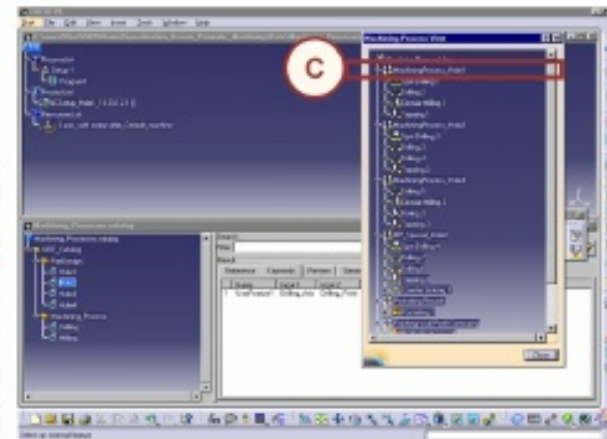
Rudiments: Storage in Catalogs and Reuse of Machining Processes (2/3)

You can add hyperlink to your Machining Process

- ◆ **Go in Knowledge advisor workbench**
- ◆ **Click "URL & Comment" icon**
- ◆ **Select your machining process**
- ◆ **Click Add button**
- ◆ **Browse to find the document (html, avi, jpg)**
- ◆ **Name the link and validate**
- ◆ **You can have more than one document**
- ◆ **Save**
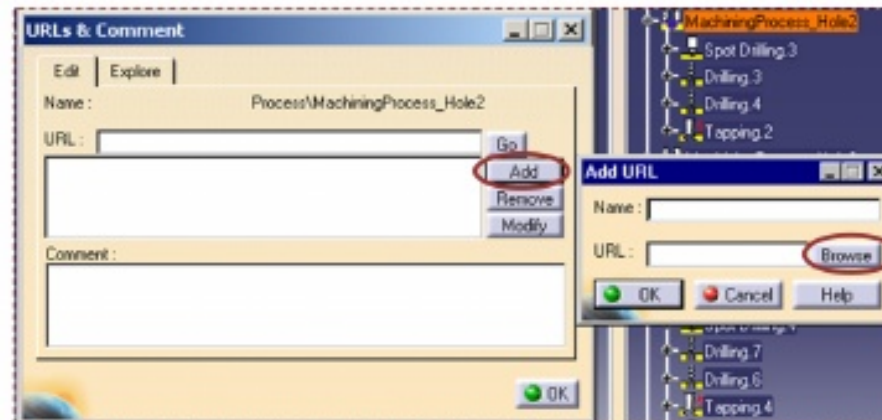
Rudiments: Storage in Catalogs and Reuse of Machining Processes (3/3)

METHOD 1
- Use Open Catalog icon to retrieve a process stored in a Catalog.
  - Click open catalog icon
  - Select your catalog
  - Select your Machining process
  - You can access to linked document
  - Define the geometry (optional)

    If you don't select geometry, after instantiation you will open each operation and select it manually
  - Select the level of insertion in the Process
  - Validate
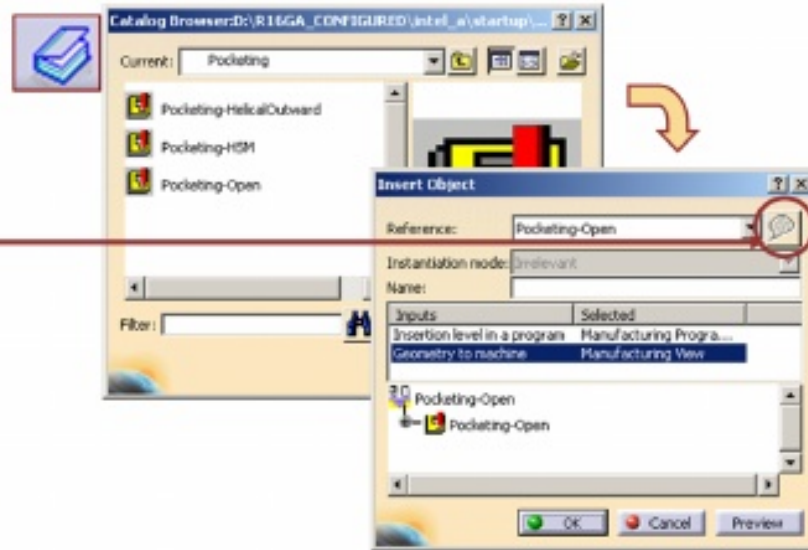
METHOD 2
- Use Machining Processes instantiation manager
  - Select your catalog
  - Select your geometry to machine (you can select more than one feature)
  - Select the level of insertion
  - Select your Machining process (you can apply more than one Machining process on your geometry
  - Validate
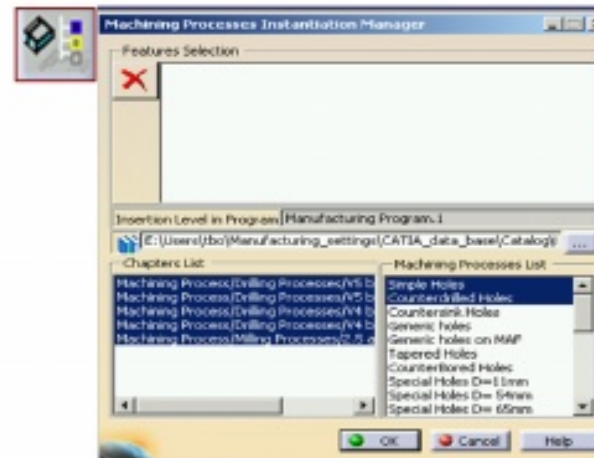
Manage Resources

In this lesson, you will learn how to create a Tools Catalog and how to add Tools from Resources in the Tool Catalog.

Also you will learn how to associate D5/V5 Machine tool.

- What is Resource Management
- Tool Catalog Management
- Need of Tools Catalog
- Create a Tool Catalog
- Create a Form Tool (User Representation)
- Associate a Machine Tool

What is Resource Management

Resource Management is effective utilization of resources while performing a Machining Operation.
The main Resources are:
- Cutting Tool and
- Machine Tool

Managing Cutting Tools means defining a Tool Catalog or creating a Tool Catalog or Updating a Tool Catalog according to availability of tools at shop-floor.

Managing Machine Tool means its association with part operation to simulate the Machine tool motions or generate transition path automatically.

You will learn about the generalities of Resource Management:
- Different processes to create a Tool Catalog
  - from Excel File
  - from Resource List
- Associate an User representation
- Add User Parameter for Tool definition
- Associate Machine tool

Tool Catalog Management

Tool Catalog Management is a major part of the Resource Management.

It consists of:
- **Generate tool Catalog**
- **Create tool Assembly in CATIA**
- **Customizing tool parameters**
- **Update existing tool catalog from ResourceList**

Why Do You Need a Tools Catalog

The Tools Catalog is the way to store the tools available in the shop floor.
These catalogs are defined under Excel Sheets and converted under catalog format
using a Visual Basic macro.
You can also update your Tool Catalog or create a new Tool Catalog from the Resources
List

- A Tools Catalog can include all tool types
  (Drills, End Mills, Taps, Conical Mills,etc)

- During a CATIA V5 session, you can access several
  Tools Catalog during operations creation in a single
  Part Operation

- In the Tools Catalog, only the cutting part of the tool
  is defined in the current version of CATIA V5
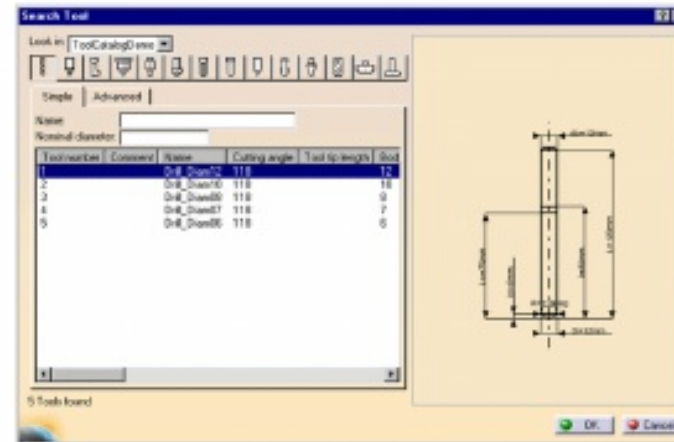
General Process to Create a Tools Catalog (1/3)

There are two different ways to create or update a tool catalog:
1- From a Excel File
2- From CATIA V5 ResourcesList (easier)

1. From a Excel File:

1

Excel Sheet with Tools description

MyCatalog.xls

Edit | Modify

Save as.... CSV type

2

CSV file with Tools description

MyCatalog.CSV

Store the MyCatalog file in the code

3

Catalog file MyCatalog.catalog Available in CATIA

Execute VB Macro

The VB Macro (MyCatalogVB2.CATScript) is available in the following directory:

..\intel_a\startup\Manufacturing\Sample
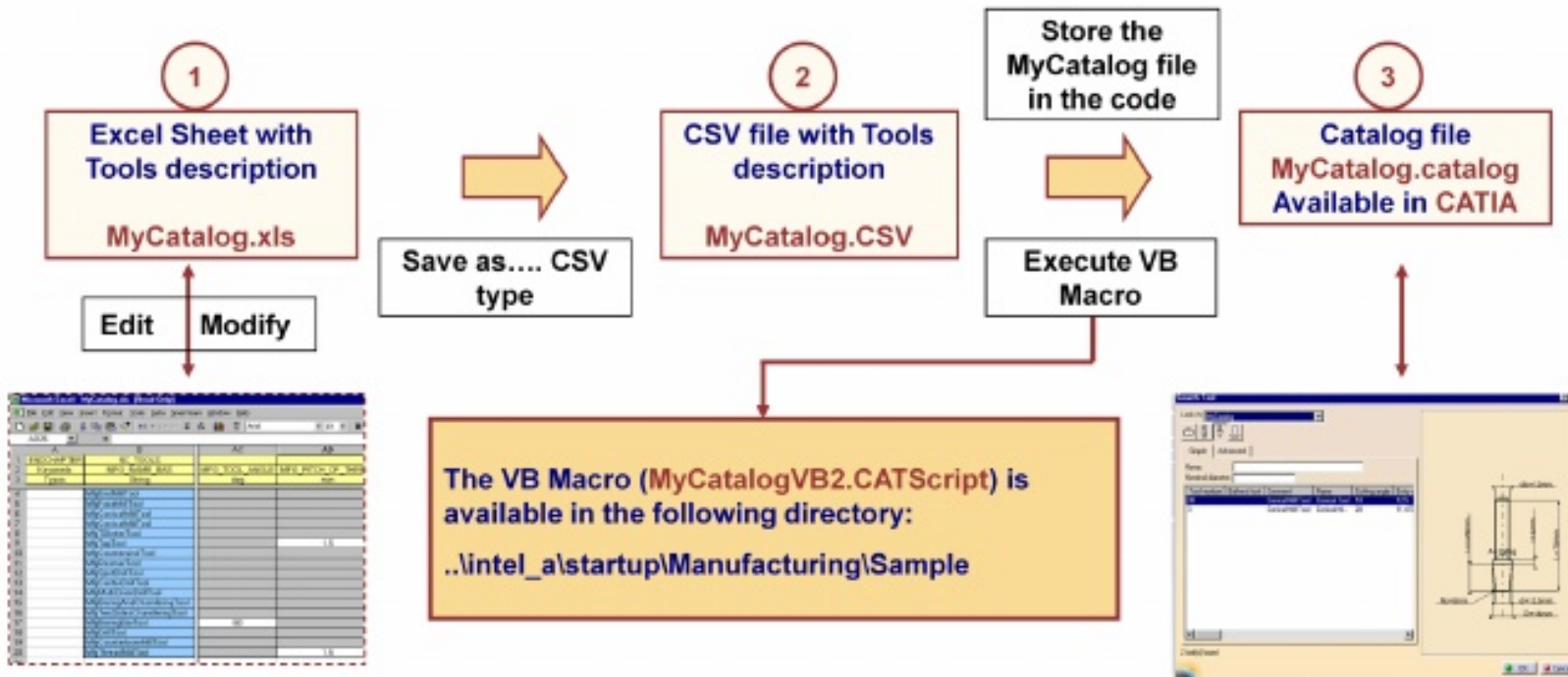
General Process to Create a Tools Catalog (2/3)

2. From CATIA V5 Resources List (1/2):

1

**CATIA V5 CATProcess
PPR tree ResourcesList**

You have defined new tool interactively in CATIA.
You use these tools in your Manufacturing Program.
1- You want to create a specific tool catalog for this Process
2- You want to update your tool catalog with these new tools

**Select the tools in the PPR Tree
MB3 and select "Send to Catalog..."**
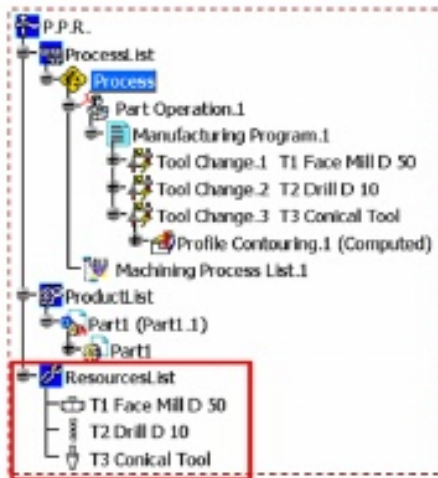
**Create a new catalog
Or
Select an existing catalog**

2

**MyCatalog.Catalog update
or
NewCatalog.Catalog creation**

You have created a new catalog or updated your catalog
The new tools are directly accessible in CATIA

General Process to Create a Tools Catalog (3/3)

2. From CATIA V5 Resources List (2/2):

From your CATIA Version 5 session:

(1) Select your tools in the ResoucesList in the PPR Tree

(2) Using Contextual menu on these tools, select "Send to Catalog …"

(3) Click "…" button to add these tool to a existing Tool catalog
Or

(4) Click OK to create a new tool catalog The new catalog is created and stored in the output directory
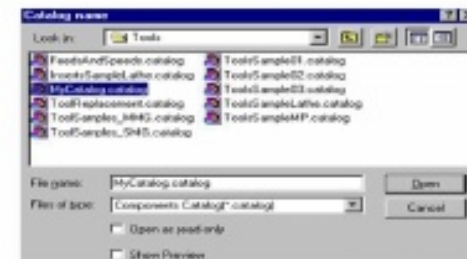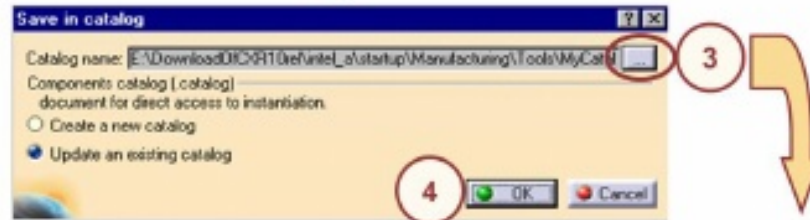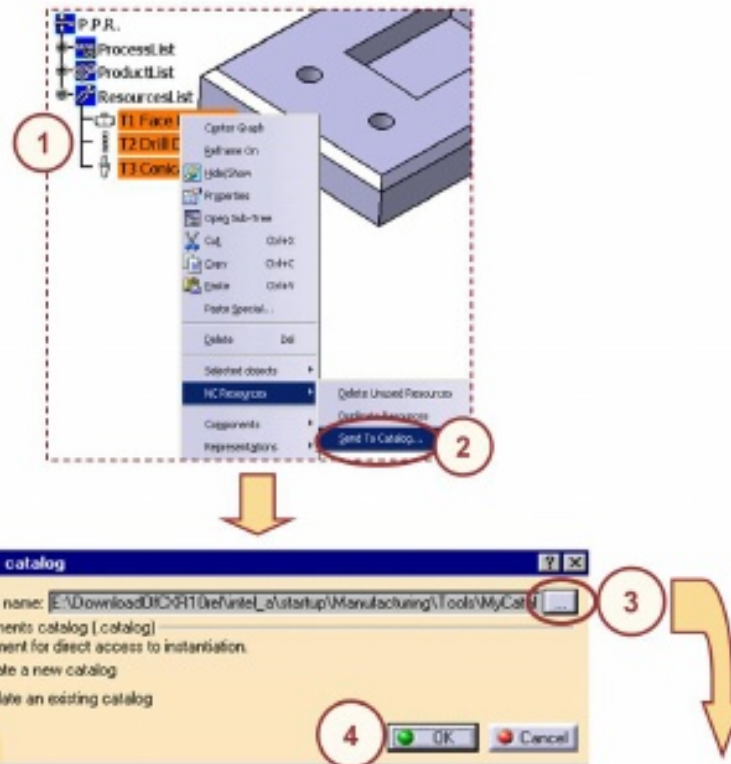
A **MyCatalog.report** file is also created in the same directory which includes a full report of the catalog creation

How to Create a Form Tool (User Representation)

Add Representation to a Tool:

1. **Create a form tool (user representation). The tool tip coordinates must be (0.0.0). Save it as a CATPart.**
2. **Associate this user representation to the tool:**
   - ◆ **From ResourceList: Via contextual menu select Add User Representation**
   - ◆ **In xls sheet: Reference is the directory where the CATPart is saved**

MFG_CORNER_RAD_2	MFG_LENGTH	MFG_BODY_DIAM	
mm	mm	mm	
0	20	15	E:\form_tool\four_cuts.CATPart

3. **This tool will be displayed in the replay of the operation.**
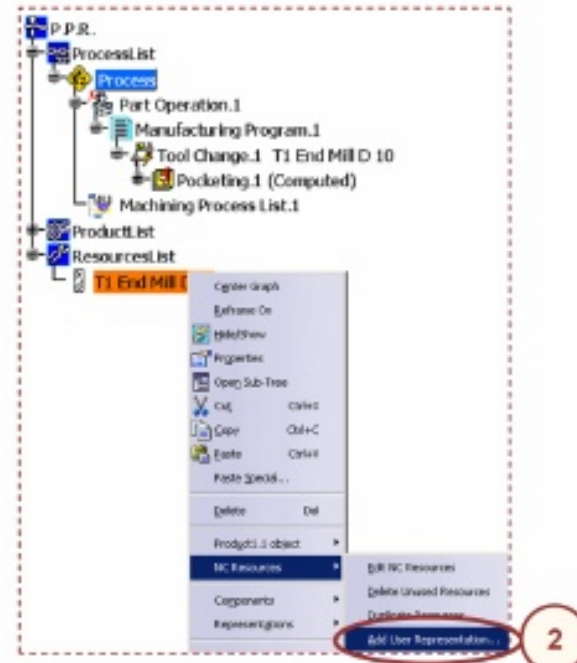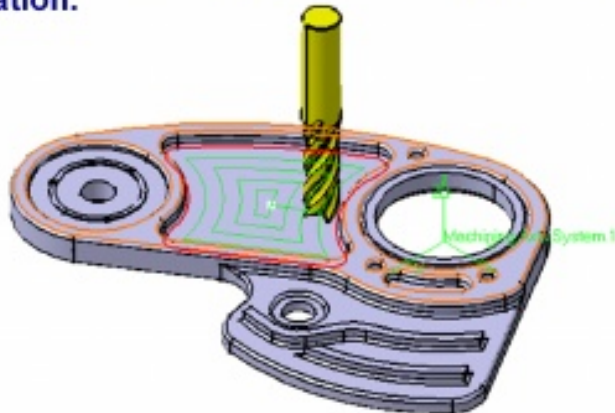
General Process to Associate a Machine Tool (1/2)

To simulate the Machine tool motions or generate transition paths automatically, you need to associate to the Part Operation, a Virtual Machine tool.(build in DELMIA product).

You will see how to do that:
1- Edit the Part operation and go in machine definition, click [icon] icon Or contextual menu on the Part operation and select « assign Machine tool from file »
2- Select the CATProduct (sample in the CATIA installation under ..\intel_a\startup\manufacturing\samples\NCMachineToollib\DEVICES)
To see how to create a machine tool, you need to follow DELMIA « machine tool builder » course
3- Put your Part/Assembly in position on the machine with auto mount functionality
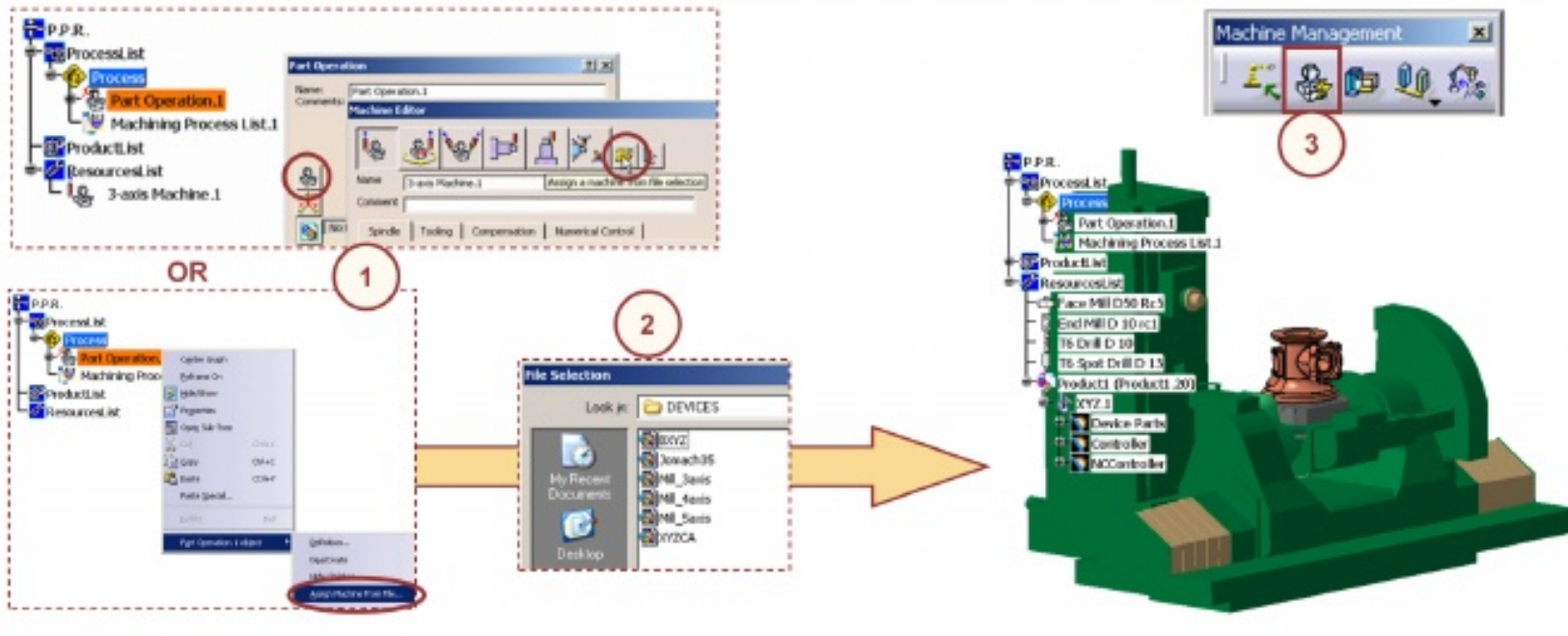
General Process to Associate a Machine Tool (2/2)

How to put in position the Part on the Machine Tool:

1. **Use automount icon to automatically put the part in position on the Machine table**
   If your Machining axis system for programming and the Machine tool axis system are the same.

   OR
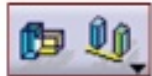
2. **Use Snap icon or Align icon to put the part in position**

   +

3. **Use attach icon to fix your NC assembly on the Machine Table.**
   Select first the Machine Table and next all the component of your NC assembly.
   Some constrains appear, you can put them in No show mode.

PP Word Table Customization

In this lesson, you will learn how to customize the Post Processor Table.

- **General Process**
- **Use of PPWord Table**

General Process of PP Word Table Customization

You can create and manage Post-Processor word tables. Each PP word table is stored in a unique text file with suffix pptable. These tables can be used as a basis for creating user-defined tables. The PP word table is associated to the machine tool in the Part Operation.

What is the use of PPWord Table

In the PP Word Table, for a given machine tool, you can define:
- NC Command: Post Processor Word syntax
- NC Instruction: Sequence of PP Word syntax

All syntaxes are stored in an unique text file with the suffix « PPTable ».

A PP word table can be defined for a specific machine tool and used in NC applications.
You can also define the general syntaxes of post-processor words. These syntaxes will be proposed when you want to create a PP instruction.

The PPWord table consists:
- **Major Word without parameters**
- **Major Word with a text**
- **Major Word with parameters**
- **Minor words**
- **Word syntaxes**

PP word Table

Design Change Management

In this lesson, you will learn how to manage Design Change.

- Introduction to Design Change
- Detection of a Modification on a Machining Operation
- Detection of a Modification on the Geometry
- Analyze the Modification on the Geometry
- Validate the Modification on the Geometry
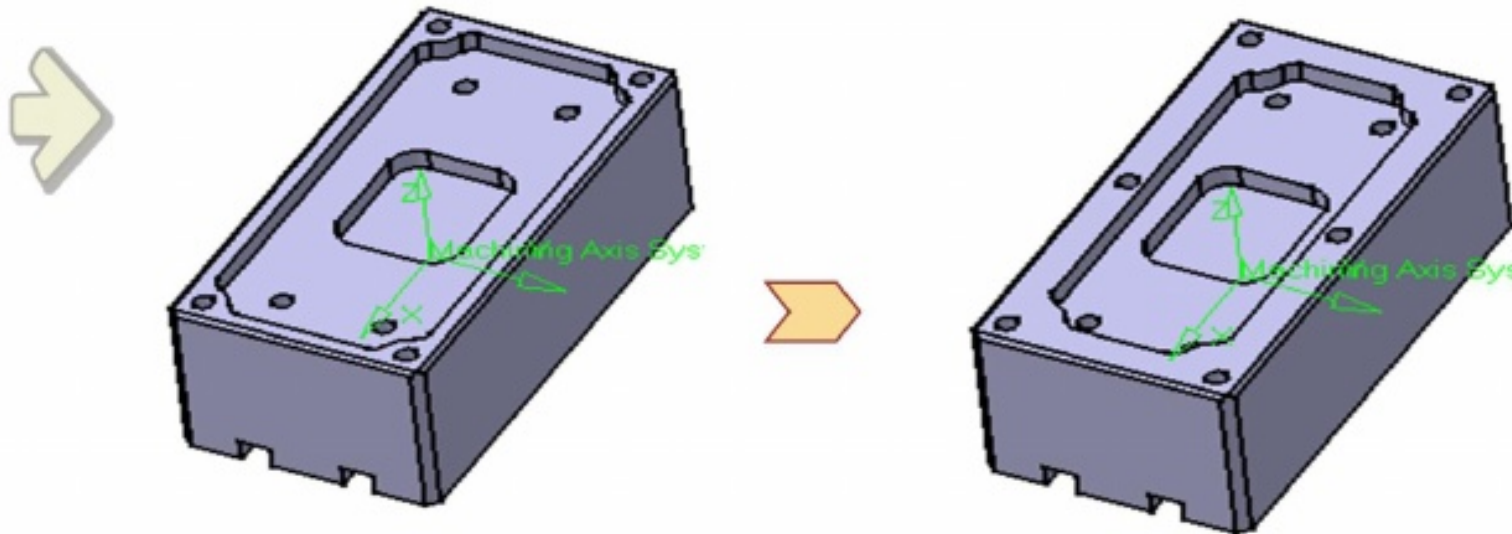
Introduction to Design Change (1/2)

Generalities:

You can see two different scenarios:

- **You have created a family of part using design table (single CATProduct)**
- **You have versioning part (different CATProduct versus1, versus2 ...)**

The methodology is similar for Machining process Update but different in the way of new geometry assignment

- ◆ **Lock your Machining operations**
- ◆ **Select the new geometry configuration**
- ◆ **Update the design**
- ◆ **Update the Machining Process**
- ◆ **Analyze the modifications**
- ◆ **Unlock your Machining operations**
- ◆ **Compute the new tool path**

To optimize design change detection you need to activate the following option:

Introduction to Design Change (2/2)

Design Modification:

- **Using Design Table:**
  - ◆ Access to you design table
  - ◆ Select the new part configuration
  - ◆ Validate

- **Using Revision Part:**
  - ◆ There is a specific methodology to create a new revision of your Part and assembly to have after a minimum of interaction in the manufacturing
    - ▪ Open the original part (Part_v1.CATPart)
    - ▪ Make the modifications
    - ▪ Save as of your CATPart (Part_v2.CATPart)
    - ▪ Open your original Product (NC_Assembly_v1.CATPRoduct)
    - ▪ Replace your original part by the new one
    - ▪ Save as of your CATProduct(NC_Assembly_v2.CATPRoduct)
  - ◆ In the part operation select the new product and validate

Detection of a Modification on a Machining Operation

After making a design change, the machining operations are displayed in the PPR tree with the following symbols:
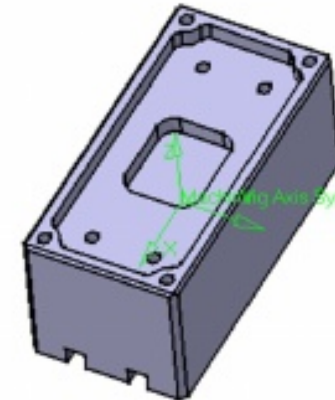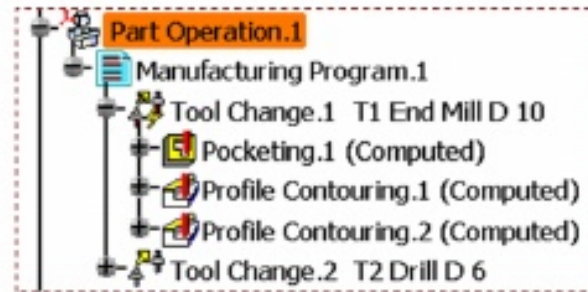
Update symbol: The geometry has changed since the last replay of the tool path

Incomplete symbol: The geometry is now missing

After Design Changes
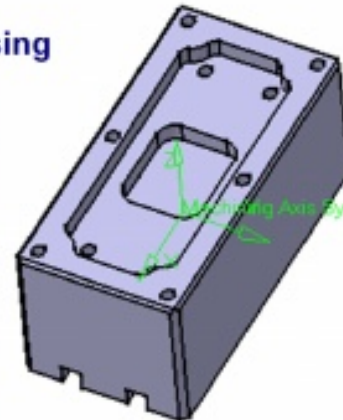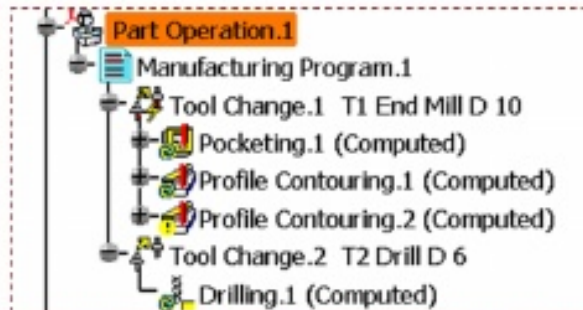
How to Detect a Modification on the Geometry

(A) **Edit a non-updated operation**

(B) **By default:**
- ◆ **a brown color indicates that the geometry must only to be updated (compute again)**
- ◆ **a purple color means that some geometry is missing to compute again the operation**

(C) **Select Analyze contextual command in the sensitive icon zone**

The Geometry Analyzer dialog box displays the status of each geometrical element associated to the Machining Operation:
- ◆ **Drive elements**
- ◆ **Check elements**
- ◆ **Relimiting planes (Top / Bottom)**
- ◆ **Relimiting elements (Start / End)**

The different status of the geometry can be
- ◆ **Up to date   --> OK**
- ◆ **Not up to date --> Operation must be replayed**
- ◆ **Not found   --> The geometry has been deleted**

The Geometry Analyzer Dialog box is displayed

Analysis of the Modification on the Geometry

Two types of analysis can be performed using the Geometry Analyzer dialog box:

Smart icon is used to visualize the original geometry used by the operation before the modification. This geometry is visualized in Red on the Part

Highlight icon is used to visualize the specified geometry used by the operation since the last modification. This geometry is displayed in Blue on the part

To remove specified geometry
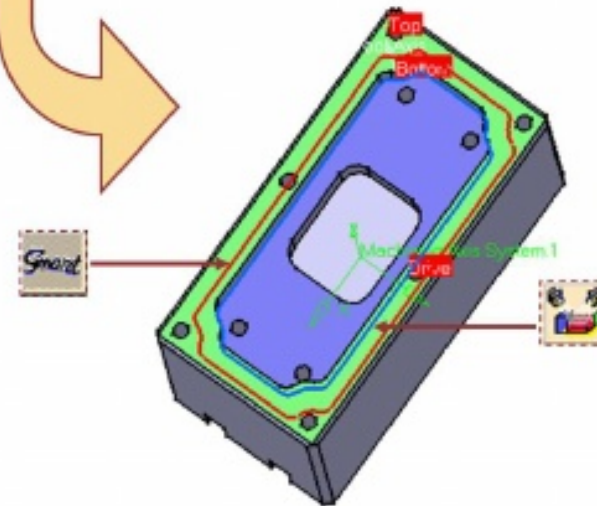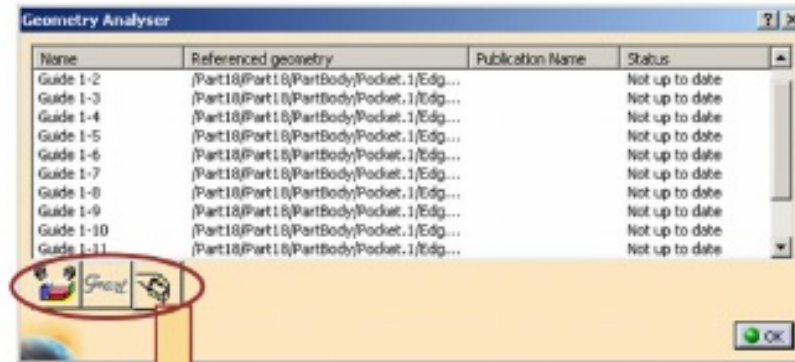
Validation of the Modification on the Geometry

To validate the operation on the new geometry and to change the status of the geometrical elements from Not up to date to Up to date, you need to Replay the tool path of the operation.

After this simulation, the operation is now consistent with the design changes and the machining operation status is also updated in the PPR graph

Name	Referenced geometry	Publication Name	Status
Guide 1-2	/Part18/Part18/PartBody/Pocket.1/Edg...		Not up to date
Guide 1-3	/Part18/Part18/PartBody/Pocket.1/Edg...		Not up to date
Guide 1-4	/Part18/Part18/PartBody/Pocket.1/Edg...		Not up to date
Guide 1-5	/Part18/Part18/PartBody/Pocket.1/Edg...		Not up to date
Guide 1-6	/Part18/Part18/PartBody/Pocket.1/Edg...		Not up to date
Guide 1-7	/Part18/Part18/PartBody/Pocket.1/Edg...		Not up to date
Guide 1-8	/Part18/Part18/PartBody/Pocket.1/Edg...		Not up to date
Guide 1-9	/Part18/Part18/PartBody/Pocket.1/Edg...		Not up to date
Guide 1-10	/Part18/Part18/PartBody/Pocket.1/Edg...		Not up to date
Guide 1-11	/Part18/Part18/PartBody/Pocket.1/Edg...		Not up to date

In case of Geometry Not Found, you need to reselect a new geometry or delete the operation if it is no longer useful.

Appendix

You will learn how to customize Machining Global Options.

- **Machining Setting: Introduction**
- **Accessing the Machining Settings**
- **Customize Settings for Machining**

Machining Setting: Introduction

In this lesson, you will learn how to customize the Machining Workbench.

Machining Settings (Tools / Options / Machining Menu)
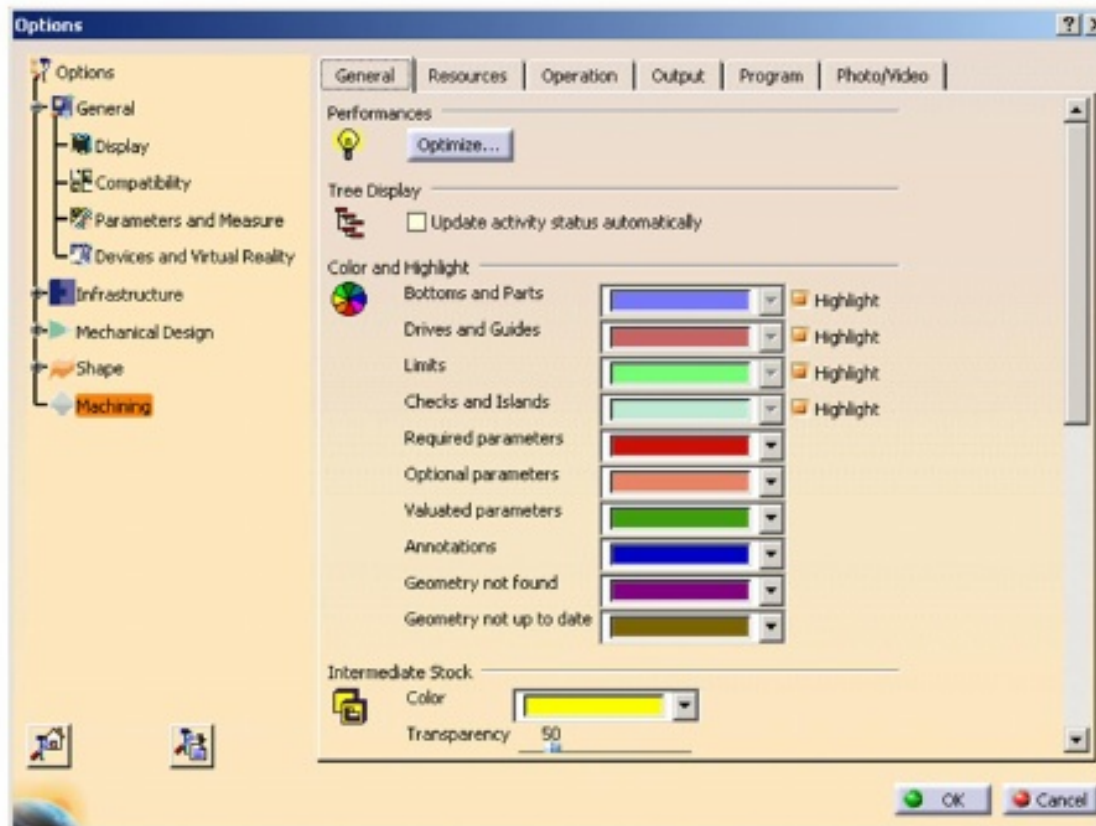
Accessing The Machining Settings

To access the Machining Settings select Tools / Options/ Machining

Customize Settings for Machining: General (1/4)

| General | Resources | Operation | Output | Program | Photo/Video |

Performances

Optimize...
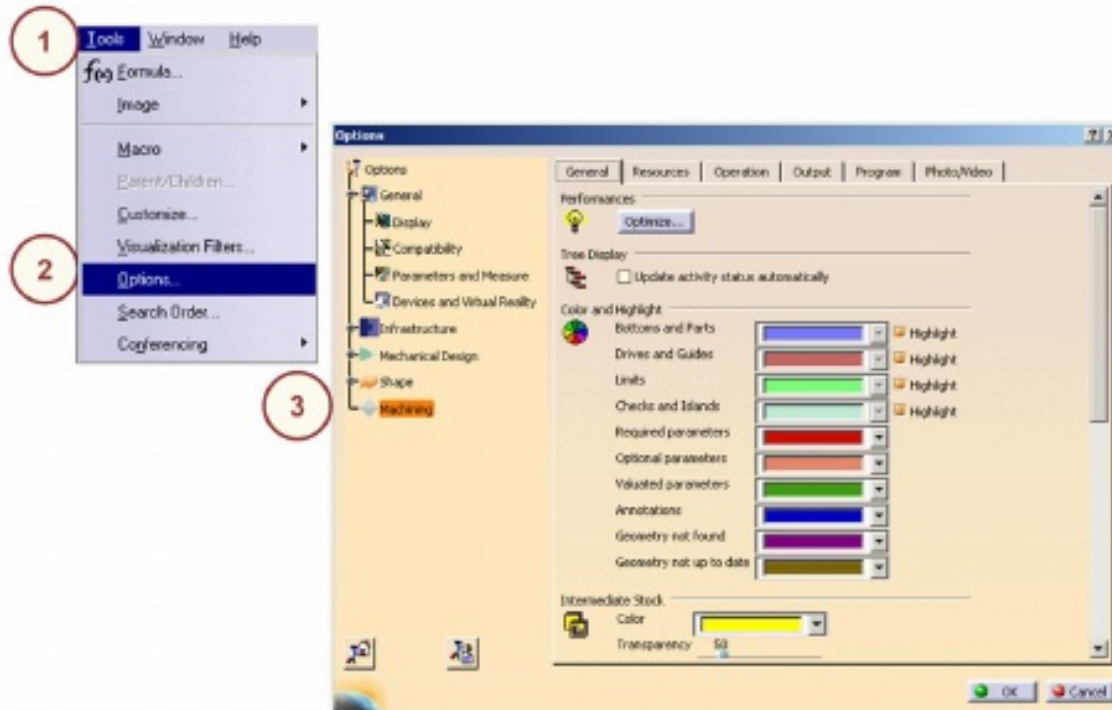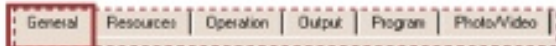
•Set by default your NC Parameters for best performances

Tree Display

☐ Update activity status automatically
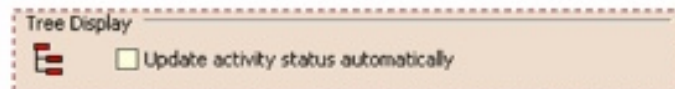
•Select this option to Update automatically your operation. (for example after design change)

Customize Settings for Machining: General (2/4)

General | Resources | Operation | Output | Program | Photo/Video

Color and Highlight

Bottoms and Parts	☐ Highlight
Drives and Guides	☐ Highlight
Limits	☐ Highlight
Checks and Islands	☐ Highlight
Required parameters	
Optional parameters	
Valuated parameters	
Annotations	
Geometry not found	
Geometry not up to date	
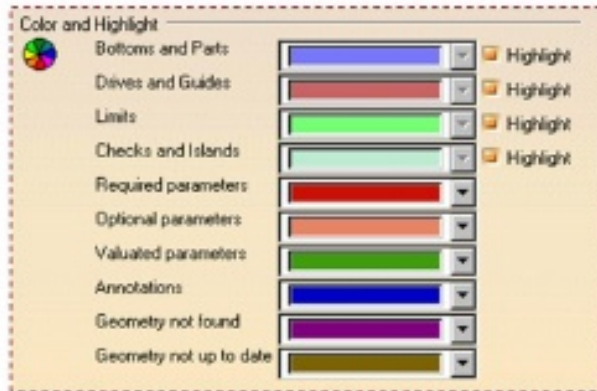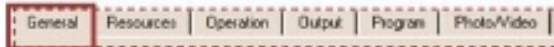
Checks and Islands

Required Parameters This color is used in all Machining Operations sensitive icon to highlight the geometry that must be selected (Bottom Plane, Drive Elements,etc)

Optional Parameters This color is used also in sensitive icon to highlight the optional geometry that can be selected (Start Elements, Top Plane, Islands,etc)

Valuated Parameters This color is used to replace the 2 previous one as soon as a geometrical element has been selected

Annotations This color is used in all the Machining Operations Geometry Tab Page to show the selectable item except the sensitive icons (Offset on Top, Stop: To/On/Past,etc)

Geometry not Found: This color is used in all Machining Operations sensitive icon to highlight the missing geometry after design change (Bottom Plane, Drive Elements,etc)

Geometry not up to date This color is used in all Machining Operations sensitive icon to highlight the geometry that must be updated after design change (Bottom Plane, Drive Elements,etc)

Limits
(Top Plane, Start Element,etc)

Bottom

Drives & guide

Offset on Check : 0mm   Offset on Top : 0mm

Required Parameter

Optional Parameter

Valuated Parameter

Annotation

Bottom : Hard

Customize Settings for Machining: General (3/4)

| General | Resources | Operation | Output | Program | Photo/Video |

Intermediate Stock

Color [yellow]
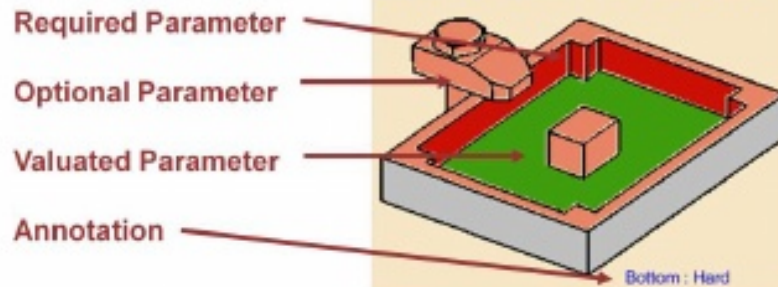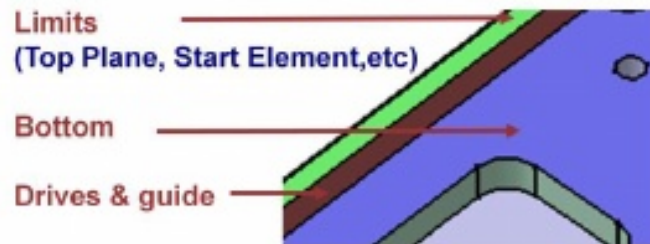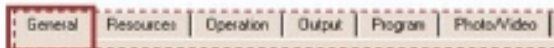
Transparency 50

◆ **Set the color and transparency for the intermediate stock**

Tool Path Replay

☐ Display tool near cursor position on tool path.

☐ Display tool center point instead of tool tip.

☑ Display circles.

Color of feedrates :

Machining (default)	[green]
Approach or Lead-in	[yellow]
Retract or Lift-off	[blue]
Rapid	[red]
Finishing	[cyan]
Chamfering	[cyan]
Plunge	[magenta]
Air Cutting	[magenta]

◆ **Select this option if you want to see the tool on the tool path. Use the mouse to indicate the position on the tool path.**

◆ **Select this option if you want to use center point to replay tool path.**

◆ **Select this option if you want to see Circle motion on the tool path**

◆ **Color Feedrates: customize the color you would like to see during tool path Replay**

Customize Settings for Machining: General (4/4)

General	Resources	Operation	Output	Program	Photo/Video

Complementary Geometry
☐ Create a CATPart to store geometry.
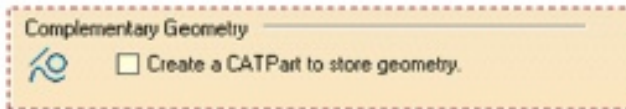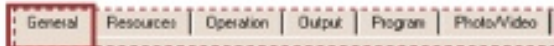
If you select this option, when you access the Manufacturing workbench with a CATPart, the system will automatically create a CATProduct with a new CATPart named « Geometry.CATPart » in which you will be able to store geometry for Manufacturing (Stock, safety plane,etc)

Design Changes
☐ Smart NC mode
☐ Optimized detection of design changes
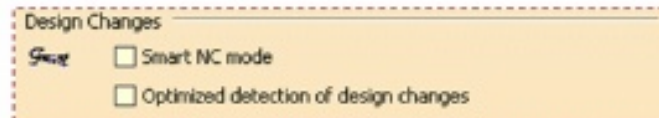
◆ **Enable Smart NC Mode** Activate this option if you want to be able to see former contour of operation after design modification. Deactivating it will save memory.
◆ **Optimized detection of design changes:** In case of Design change by replacing Product (edit links or PO Product association). Allow to detect Identical element (Mathematic Comparison) to reduce element to reselect in the Machining operation

Customize Settings for Machining: Resource (1/2)

General | **Resources** | Operation | Output | Program | Photo/Video |

Catalogs and Files for Tools, PP Tables, Machining Processes and PPs

E:\DownloadOfCXR10ref\intel_a\startup   [...]
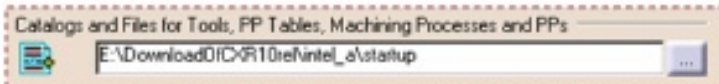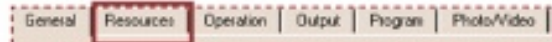
The Location Path is used to reference the directory that includes all the Tools Catalogs, Machining Processes, Post Processors and the PP Words Tables necessary during Machining Operations creation.

Under this Location Path, the following structure is mandatory:

Manufacturing\Tools	⟶ **for Tools Catalogs**
Manufacturing\PPTable	⟶ **for PP Words Tables**
Manufacturing\Processes	⟶ **for Machining Processes**
Manufacturing\IMSPar	⟶ **for IMS Post Processors**

The default location used by the software is:
C:\Program Files\Dassault Systemes\B05\intel_a\startup

Tool Selection
- ☑ Automatic query after modification
- ☑ Tool preview after selection
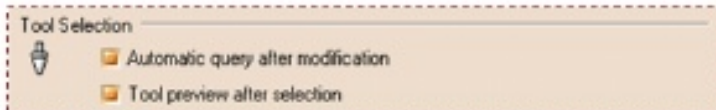
♦ **The Automatic Query after Modification check box deactivation avoids to search in the Tools Catalog each time a modification is performed on a Tool attribute. The query will be done only on user choice. This is an important point for performance when you have a huge Tools Catalog.**

♦ **The Tool Preview after Selection check box activation allows to display the graphic representation of the tool in the Search Tool dialog box**

Customize Settings for Machining: Resource (2/2)

| General | Resources | Operation | Output | Program | Photo/Video |
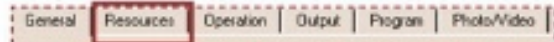
Automatic compute from Tool Feeds and Speeds
- ☑ for Feedrate attributes of the operation
- ☑ for Spindle attributes of the operation

◆ **The system automatically compute the Machine feedrates according Tool Feeds & Speeds**

◆ **The system automatically compute the Machine spindle according Tool Feeds & Speeds**

Tool Query mode in Machining Processes instantiation
- ○ Automatic Tool Query
- ◉ Interactive Tool Selection in case of multiple results
- ☐ Interactive Tool Selection if no tool is found

◆ **When you instantiate a Machining process, the system select the first tool it found in your catalog corresponding to your query**

◆ **When you instantiate a Machining process, the system asks you which tool you want in case of multiple results during the tool queries**

◆ **When you instantiate a Machining process, the system asks you which tool you want in case of no results during the tool queries**

Customize Settings for Machining: Operation (1/2)

| General | Resources | Operation | Output | Program | Photo/Video |

Default Values

▤ ☐ Use default values of the current program
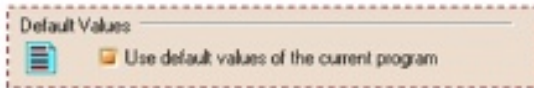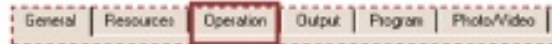
◆ **Use default values of the current program: select this option if you want operations to be created with the values used in the current program. Otherwise the default settings delivered with the application are used.**

After Creation or Machining Process (MP) Instantiation

☐ Sequence machining operation

☐ Search compatible tool in previous operations

☐ Use a default tool

☐ Start Edit mode (not available for MP)

☐ MP instantiation: keep absolute position of tool axis and machining directions
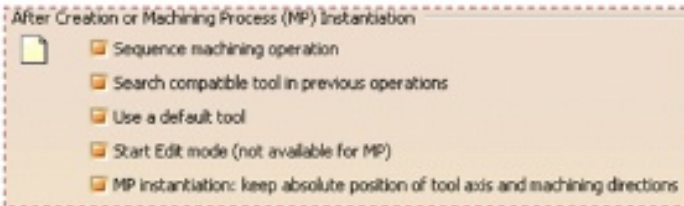
◆ **Sequence machining operation after creation: deactivate this option if you want to create operation in Manufacturing View window.**

◆ **Search compatible tool in previous operation of the current program after creation: If this option is activated, the system will look for a compatible tool in the current Manufacturing Program during Machining Operation creation**

◆ **Use a default tool: If this option is activated, the system will look in the Resources List to find a compatible tool already used in any Manufacturing Program.**

◆ **Start Edit mode just after creation: The activation of this option allows to edit the operation for geometry selection at the creation step. The deactivation of this option will create Not Complete operation which must be edited after to select the missing geometry**

◆ **MP instanciation: keep absolute position of tool axis and machining directions: Allows the user to manage the absolute or relative mode for the machining direction when a machining process is instantiated. The user can instantiate the MP with respect to the default absolute axis system or to the current axis system for the machining direction. It can be managed like the tool axis while instantiating a machining process**

Customize Settings for Machining: Operation (2/2)

| General | Resources | Operation | Output | Program | Photo/Video |

When Copying

☐ Duplicate geometry links
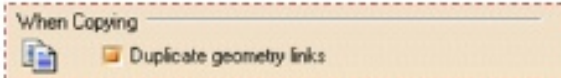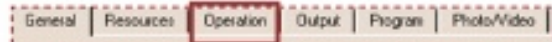
◆ **Duplicate geometry links when copying**
During the Copy / Paste of an Operation, you will copy also the links with the geometry if the option is activate. In the other case, the Operation will be copied without geometry and with the Not Complete status

Display

☐ Display tool path of operation in current Part-Operation
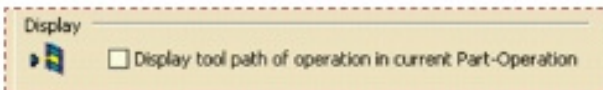
◆ **Display tool path of operation in current part operation**
Keep the tool path display on the screen. (you can put it in no show mode)

User Interface

☐ Simplified mode

◆ **Simplify user interface:** Hide parameters in Machining operation definition
(available only in SMG-MMG)

When Reversing Operation
Reverse Keyword -reversed

◆ **When Reversing Operation:** The Reverse keyword defined here will be appended to the name of Machining Operations which are reversed using Reverse Machining Condition function. The PPR tree is updated with the operations name modified after displaying a pop-up message.

Customize Settings for Machining: Output (1/2)

| General | Resources | Operation | Output | Program | Photo/Video |

Post Processor

● None ○ Cenit® ○ IMS® ○ ICAM®

PP Path: E:\DownloadDITCXR10rel\intel_a\startup\Manufacturing
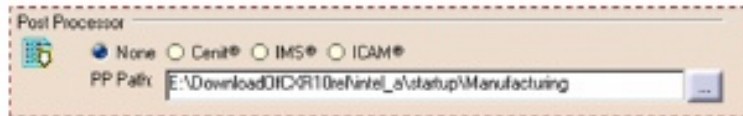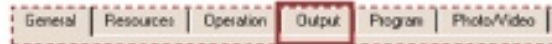
**Select in this folder the type of post processor for NC code generation.
Select the path where are stored your Post Processors
(by default it's in C:\Program Files\Dassault Systemes\B10\intel_a\Startup\Manufacturing)**

Performances and Memory Management

Tool Path Storage

● Store tool path in the current document
○ Store tool path in an external file
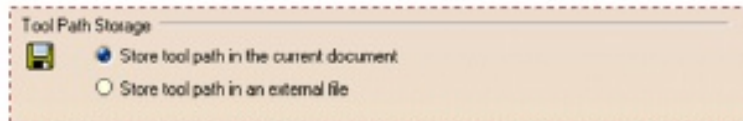
◆ **Tool Path Storage: You can choose to pack tool path on disk in a tlp file or to include it in the current
document. (default is tool tip)**

Tool Path Edition

☑ Edit Tool Path is available

◆ **Tool Path edition: You have access to the tool path to edit it and modify it. If you want to make
modification you must before lock the machining operation (think contextual menu on it)**

Customize Settings for Machining: Output (2/2)

| General | Resources | Operation | Output | Program | Photo/Video |

During Tool Path Computation
☐ Store contact points in tool path

During Tool Path Computation: activate this option if you want to store contact points in tool path during tool path computation

Tool Output Point
◉ Tool Tip
○ Tool Center
○ Tool Center for Ball End Tools

Tool Output Point: Select the Output Tool Point you want to manage, the NC File will be generated according with this choice
- ◆ **Tool Tip: always generate tool tip coordinates**
- ◆ **Tool center: always generate tool center coordinates**
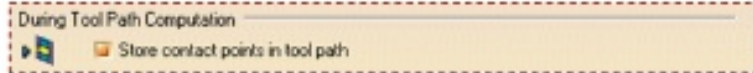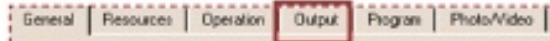- ◆ **Tool center for ball end: Generate tool center output coordinate only for any tool with « ball end » attribute or with Nominal diameter equal to 2x Corner radius, for the other tool, generate Tool tip coordinates**

Tool Path files, NC Code output and NC Documentation Location
Tool path: ☐ Store at the same location as the CATProcess
C:\DOCUME~1\YUF\LOCALS~1\Temp\
Video result: ☐ Store at the same location as the CATProcess
C:\Documents and Settings\YUF\Desktop
NC Doc: C:\DOCUME~1\YUF\LOCALS~1\Temp\
NC Code: C:\DOCUME~1\YUF\LOCALS~1\Temp\
Extension: CATNCCode
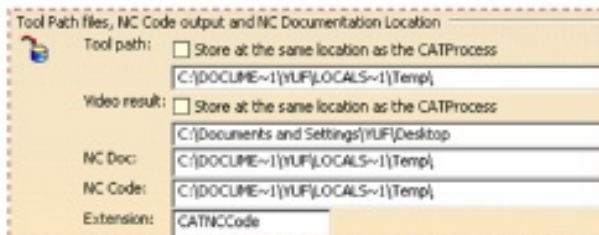
Tool Path Location: Directory where the file corresponding to the
- ◆ **Tool path**
- ◆ **NC Doc**
- ◆ **NC Code will store.**

During NC Code Generation
☐ Do not generate NC Code if MO is not uptodate
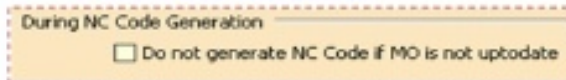
During NC Code Generation:

Checking on this it would not generate NC Code if MO is not updated.

Customize Settings for Machining: Program

| General | Resources | Operation | Output | Program | Photo/Video |

Auto Sequencing

☐ Access to sequencing rules settings
Sequencing rules path

C:\Program Files\Dassault Systemes\B09\intel_a\startup\Manufacturing ...

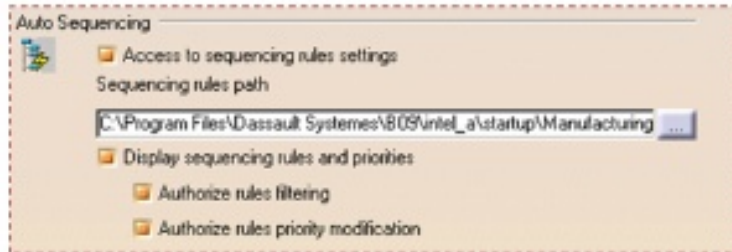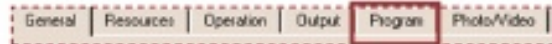☐ Display sequencing rules and priorities

☐ Authorize rules filtering

☐ Authorize rules priority modification

Sequencing rules path: Path where is store the file containing Sequencing rules
Make sure that the document in the sequencing rules path (AllSequencingRules.CATProduct in the example above) is accessible in Read-Write.

The settings in the Auto-Sequencing area are mainly intended for the administrator

◆ **Access to sequencing rules settings: Select the first check box to authorize user access to sequencing rules**
◆ **Display sequencing rules and priorities: Select the second check box to authorize the display of sequencing rules and priorities in the user's view. In this case two more check boxes can be selected in order to:**
  • **Allow the user to filter rules**
  • **Allow the user to modify rule priorities**

Customize Settings for Machining: Photo/Video (1/2)

General | Resources | Operation | Output | Program | Photo/Video

Simulation at
- ● Program level ○ Part operation level
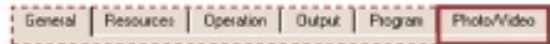
◆ **select the required option to perform tool path simulation at either Program or Part Operation**

Video
- ☐ Stop at tool change
- Collisions detection : ○ Ignore ○ Stop ● Continue
- ☐ Touch is Collision
- ☐ Multiple Video result on program
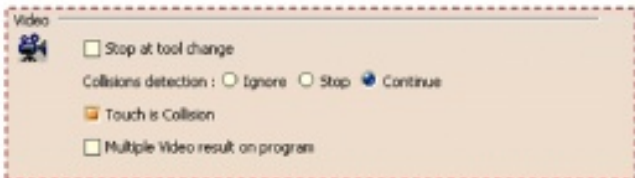
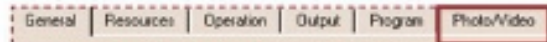◆ **Stop the simulation at each tool change (press forward button to continue the simulation)**
◆ **In the Video Collision area, select the required option to:**
  - **Ignore collisions during the Video simulation**
  - **Stop the Video simulation at the first collision**
  - **Continue the Video simulation. In this case, you can consult the list of collisions at the end of the simulation.**
◆ **Select the Touch is collision check box if you want that type of collision to be detected.**
◆ **Select Multiple video result on program if you want to store more than one intermediate video result**

Global
- Fault box ● Wireframe ○ Transparent ○ None
- ☐ Compute all information at picked point
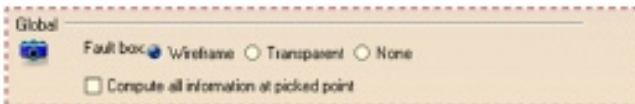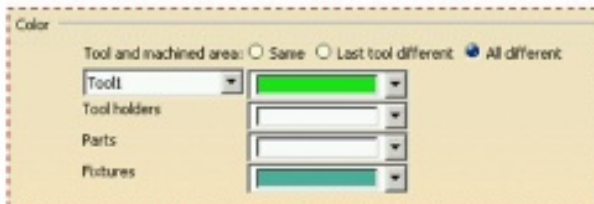
◆ **In the Fault Box area, select the required box type for examining remaining material or gouges:**
  - **Transparent: to display a transparent bounding box**
  - **Wireframe: to display a wireframe bounding box**
  - **None: if no bounding box is required.**
◆ **Select the Compute all information at picked point check box if you**

Customize Settings for Machining: Photo/Video (2/2)

General	Resources	Operation	Output	Program	Photo/Video

Performance
- Tool and faceting: ○ Smaller ○ Larger ● Standard
- Photo resolution: 0
- Tool axis interpolation angle (5 axis only): 1 deg
- ☑ Optimized rendering for Video

♦ **In the Performance area, you can:**
  - select the required option for facetization of the tool (Smaller, Larger or Standard)
  - set the resolution for Photo simulation. It can be increased from 0 to improve machining accuracy and give a very detailed simulation. However, a higher resolution results in more memory and time being consumed for the simulation.
  - specify the maximum angle that the tool axis is allowed to vary between two consecutive points
  - Optimize rendering for video: remove the rendering and the color of your part for the simulation to increase the computation performances

Color
- Tool and machined area: ○ Same ○ Last tool different ● All different
- Tool1
- Tool holders
- Parts
- Fixtures

♦ **In the Color area, you can:**
  - set the tool color to be the same as or different from the last tool, or have different colors for all tools.
  - assign colors to the different tools, the Parts, the fixtures & the holders using the color combos.

Positioning Move
- Maximum tool axis variation: 1 deg

♦ **In the Positioning Move area, set the maximum allowed tool variation in the transition path between two operations.**