STUDENT GUIDE

Student Notes:

CATIA V5 Training

Foils

Prismatic Machining

Version 5 Release 19
January 2009

EDU_CAT_EN_PMG_FF_V5R19

Student Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to:

- **Identify and use the Prismatic Machining workbench tools**
- **Define Prismatic Machining operations (2.5 Axis Milling) in CATIA V5**
- **Create Prismatic Machining Area and Rework Area**
- **Define and modify NC Macros**

Targeted audience

NC Programmers

Prerequisites

Students attending this course should have knowledge of CATIA V5 Fundamentals and Numerical Control Infrastructure workbench

16 hours

Student Notes:

Table of Contents (1/5)

Student Notes:

Table of Contents (2/5)

STUDENT GUIDE

Student Notes:

Table of Contents (3/5)

Student Notes:

Table of Contents (4/5)

Student Notes:

Table of Contents (5/5)

Student Notes:

How to Use This Course

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

Lessons:

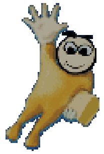
Lessons provide the key concepts, methodologies, and basic skill practice exercises. The goal of each lesson is to present the necessary knowledge and skills to master a basic level of understanding for a given topic.

A Master Exercise:

A Master Exercise provides a project where an industry scenario 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.

Added Exercises:

Added Exercises are provided after execution of Master Exercise for additional self practice.



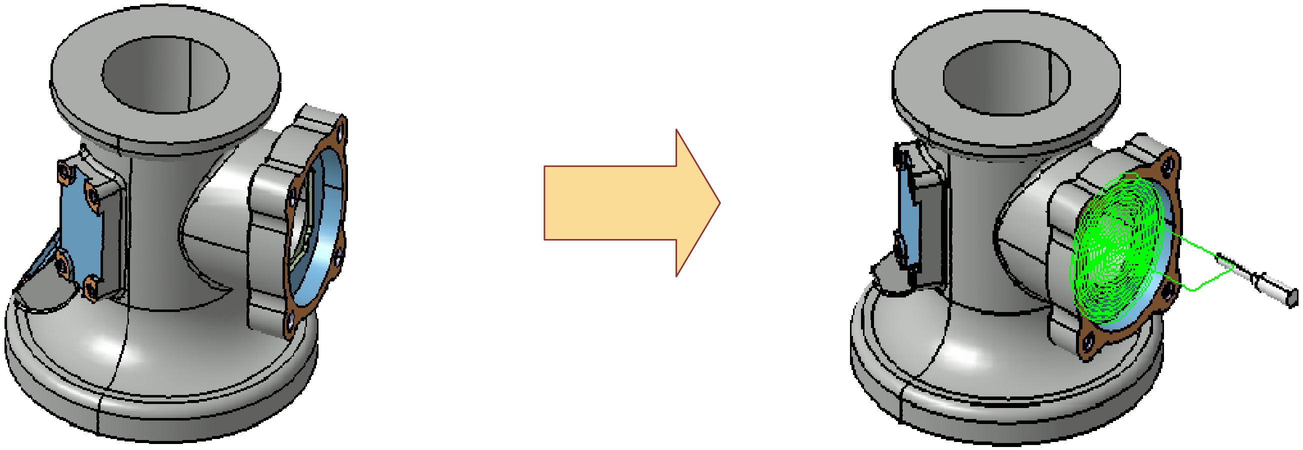
Note*: *According to preference, Master Exercise individual step may be completed after an individual lesson containing its key concepts.

Student Notes:

Introduction to Milling Operations

In this lesson, you will discover fundamentals of Prismatic Machining in Prismatic Machining Workbench.

- Introduction to Prismatic Machining
- Accessing the Workbench
- Exploring the User Interface
- Creating a Milling Operation



Copyright DASSAULT SYSTEMES

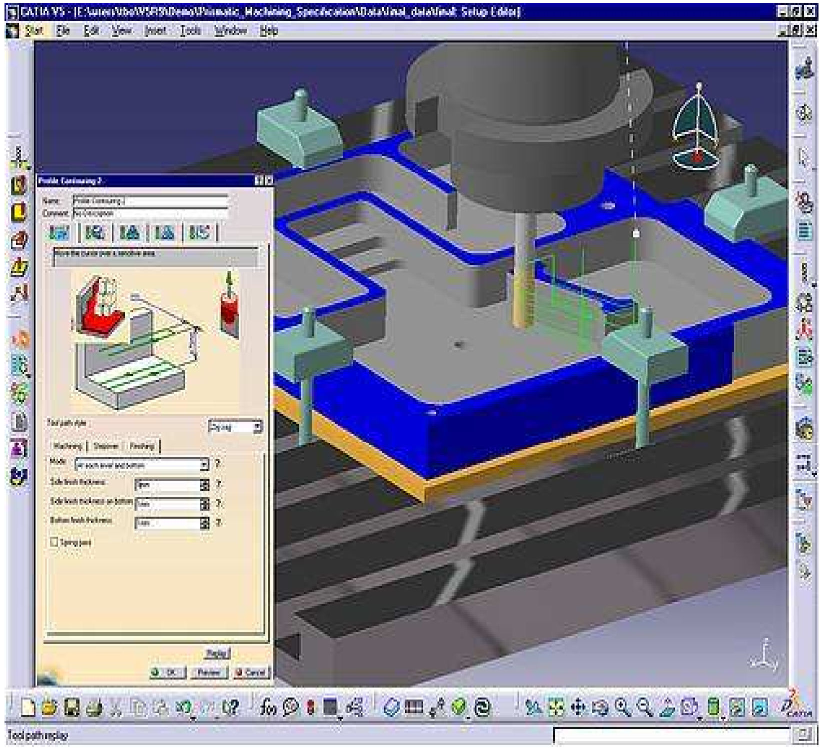
Student Notes:

Introduction to Prismatic Machining

Prismatic Machining workbench enables you to define and manage NC programs dedicated to machining parts designed in 3D wireframe or solids geometry using 2.5 axis machining techniques.

Prismatic Machining offers the following main functions:

- **2.5 axis milling and drilling capabilities**
- **Management of tools and tool catalogs**
- **Flexible management of the manufacturing program with intuitive and easy-to-learn user interface based on graphic dialog boxes**
- **Tight interaction between tool path definition, verification and generation**
- **Seamless NC data generation due to an integrated Post Processor Access solution**
- **Automatic shop floor documentation in HTML format**
- **High associative level of the manufacturing program ensures productive design change management due to the integration with Version 5 modeling capabilities**

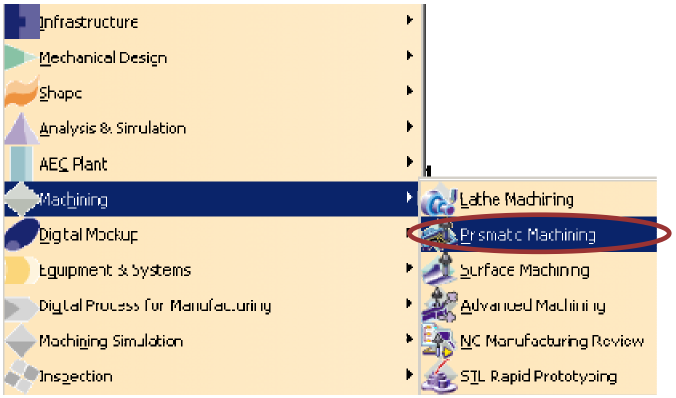


Student Notes:

How to Access the Prismatic Machining Workbench

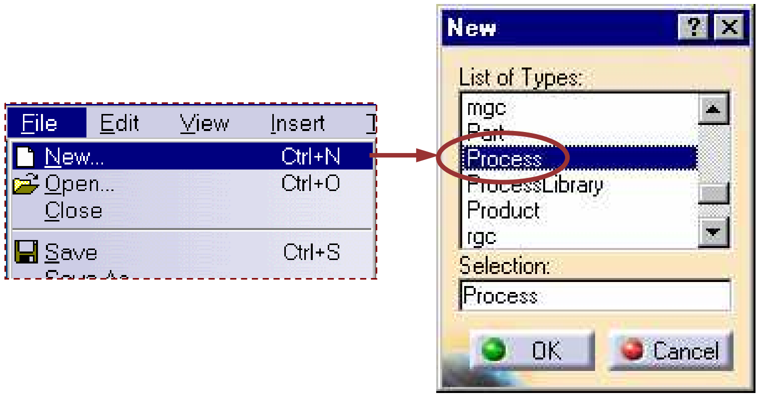
You will learn the different methods to access the Prismatic Machining workbench.

A **Start Menu**

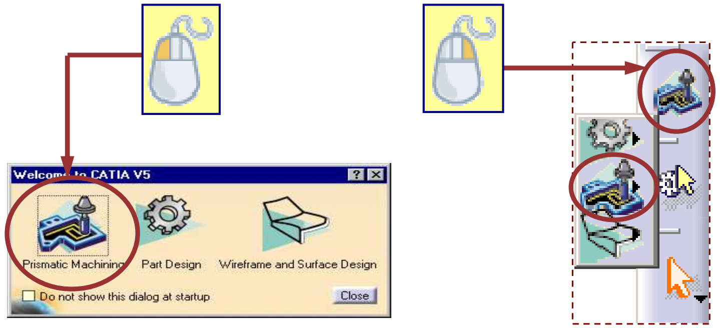


Anywhere from A - Start menu
or B - File menu + New
or C - Workbench Icon

B **New Manufacturing CATProcess to start**



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



Student Notes:

Workbench User Interface

You will become familiar with functionalities available in Prismatic Machining Workbench.

Icon	Name	Definition
	Pocketing Operation	It machines open or closed pocket with or without inner domains.
	Facing Operation	It is a plane milling operation used for cutting constant offset of material on a planer area.
	Profile Contouring Operation	It consists in cutting material along a hard boundary in same or zig-zag direction.
	Curve Following Operation	It machines a part by following a curve with the tool tip.
	Groove milling Operation	It allows you to machine groove area with a T-slot tool.
	Point To Point Operation	It consists in moving the tool from a selected point to another selected point at a given machining feedrate.
	Prismatic Roughing Operation	It is a operation used to rough machine the drafted or multiple bottom pockets.
	Prismatic Machining Area	It allows you to define an area from your geometry and record it. Further this area is used for pocketing or profile contouring.
	Prismatic Rework Area	It is the area which is remained unmachined after performing the previous operation.

Student Notes:

How to Create a Milling Operation

Milling operation defines the complete Milling process that will be followed.

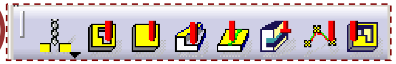
1 **Click any of the milling operation icon**

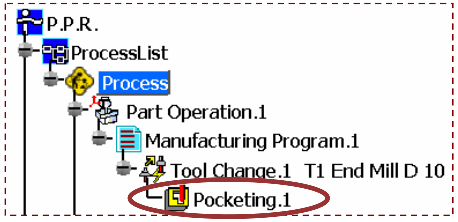
The new Operation is created after the current one. The Operation dialog box appears to edit it

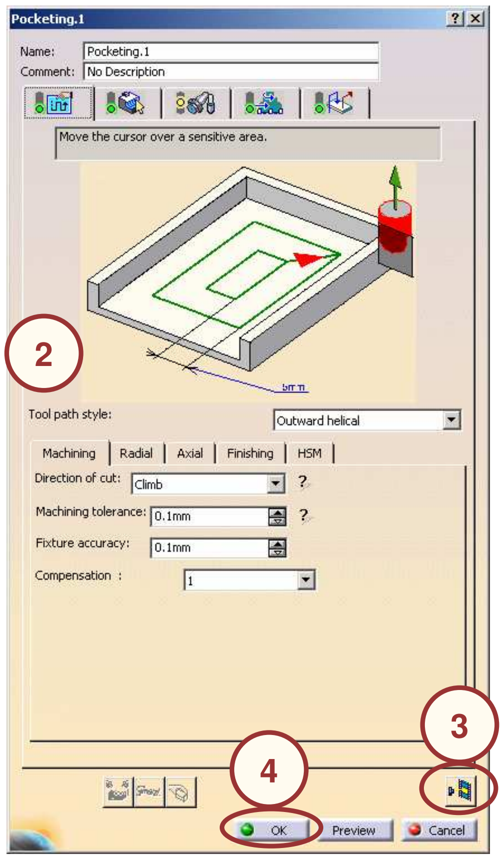
2 **Define the operation geometry and parameters in the dialog box**

3 **Replay the Tool Path**

4 **Confirm Operation creation**







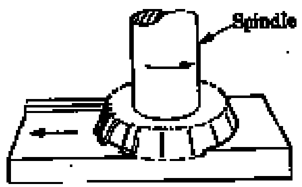
The Operation is created in the PPR tree with a default tool. This capability can be removed by customizing the NC Manufacturing options.

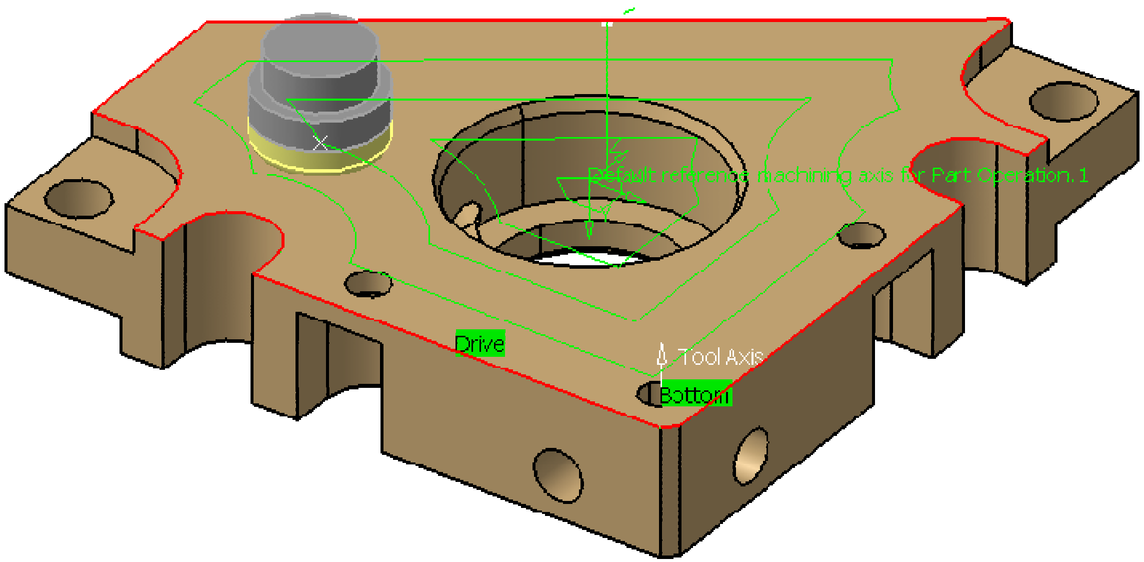
Student Notes:

Facing Operation

In this lesson, you will learn what is a Facing Operation and how to define it.

- Introduction
- Creating a Facing Operation
- Strategy
- Geometry
- Select/Create a Tool
- Feeds and Speeds





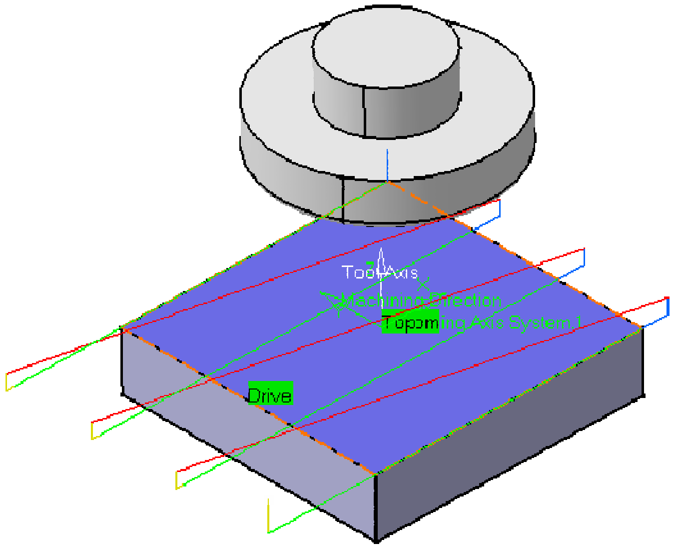
Facing Operation: Introduction

A Facing Operation consists in cutting a constant offset of material on a planar area.

Facing operation is a 2D plane milling operation used to create plane reference surfaces.

In a Facing Operation:

- **The tool axis is normal to the planar area.**
- **The material is removed in one or several cuts along the radial and axial directions.**
- **The area is machined with Inward Helical, Back & Forth or One Way tool path style.**
- **The Planar area is delimited only with soft boundaries.**



In Facing Operation, the cutter is mounted on a spindle having an axis of rotation perpendicular to the work piece surface.The milled surface results from the action of cutting edges located on the periphery and face of the cutter.

Student Notes:

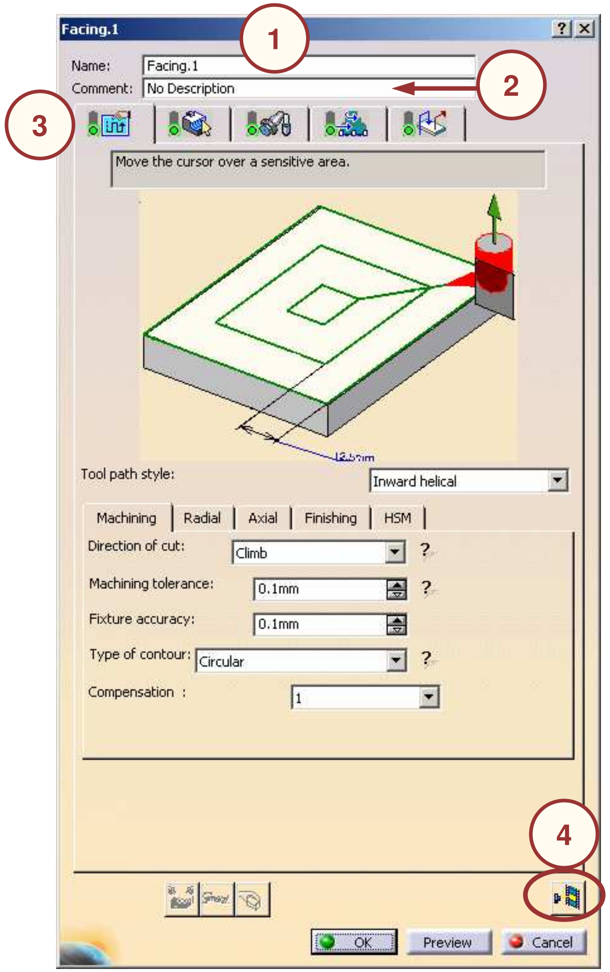
Student Notes:

Creating a Facing Operation: General Process

1. Type the Name of the Operation. (Optional because a default name is given by the system 'Type_Of_Operation.X')
2. Type the text of comment (optional)
3. Define operation parameters using the 5 tab pages

- Strategy tab page
- Geometry tab page
- Tool tab page
- Feeds & Speeds tab page
- Macros tab page

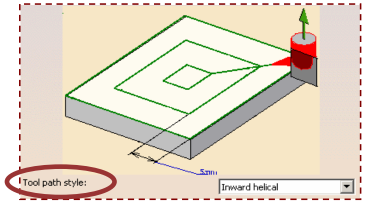
4. Replay and/or Simulate the operation tool path



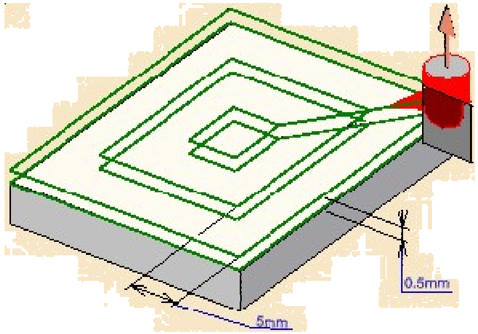
Student Notes:

Facing Operation: Strategy (1/5)

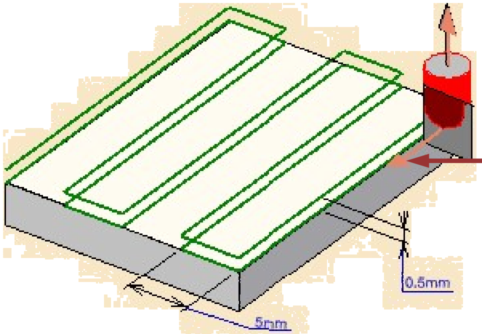
The three possible "Tool path styles" for a Facing Operation are as shown below. Inward Helical and Back & Forth are generally used for rough facing operation. One Way is used generally for finish facing operation.



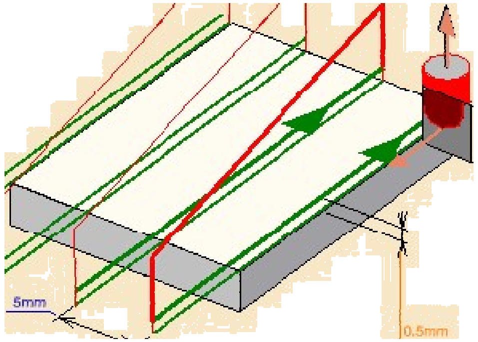
Inward Helical



Back & Forth



One Way



Inward Helical:
The tool starts from a point outside the area and follows inward paths parallel to the boundary.

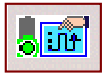
Back & Forth:
The tool alternatively machines in one direction then in the opposite direction.

One Way:
The tool always machines in the same direction.

To change the machining or progression direction select the arrow

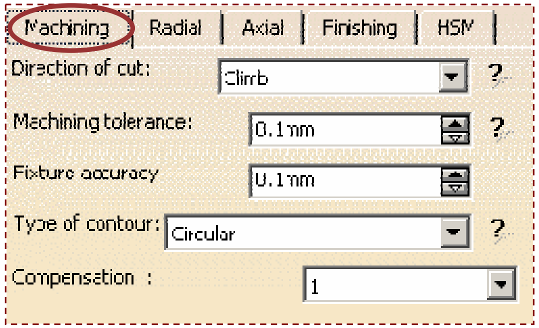
Student Notes:

Facing Operation: Strategy (2/5)

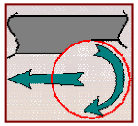


Machining Strategy Parameters:

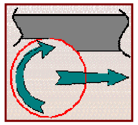
Direction of Cut
Climb: The front of the advancing tool cuts into the material first
Conventional: The back of the advancing tool cuts into the material first



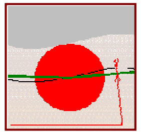
Climb Milling



Conventional Milling

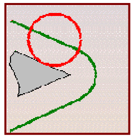


Machining Tolerance
Value of the maximum allowable distance between the theoretical tool path and the tool path computed

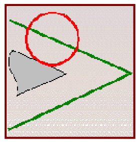


Fixture Accuracy
Local machining tolerance for fixtures

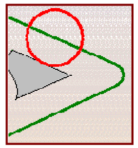
Type of Contour
Circular: The tool pivots around the corner point, following a contour whose radius is equal to the tool radius
Angular: The tool does not remain in contact with the corner point, following a contour consist of 2 line segments
Optimized: The tool follows a contour derived from the corner that is continuous in tangency
Forced Circular: The tool follows a near-circular contour consisted of line segment



Circular



Angular

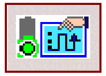


Optimized

Compensation: Number of the tool compensation used if this one is already defined on the tool

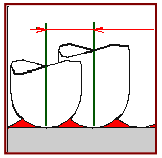
Student Notes:

Facing Operation: Strategy (3/5)

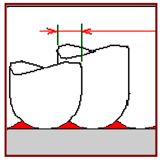


Radial Strategy Parameters:

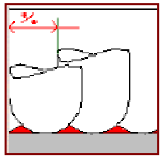
Maximum Distance
A maximum distance will be used to compute the distance between two paths

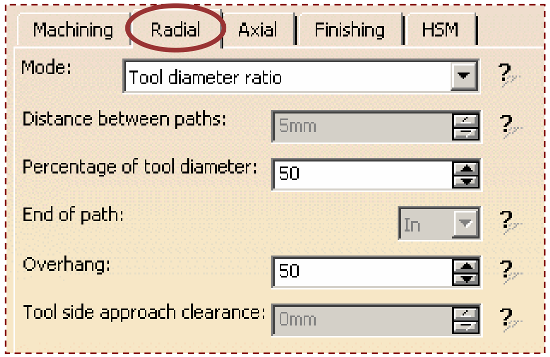


Tool Diameter Ratio
The distance between two paths will respect a tool diameter overlap ratio recovery



Stepover Ratio
The distance between two paths with respect to a stepover ratio
(10% stepover = 90% tool diameter)



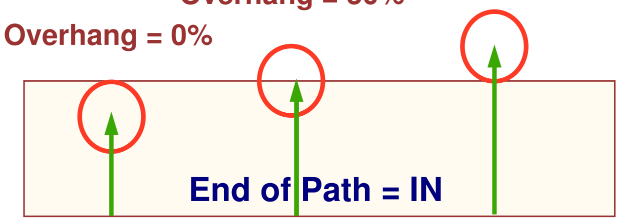


Overhang: Extension of the tool path in percentage of the tool diameter

Overhang = 100%

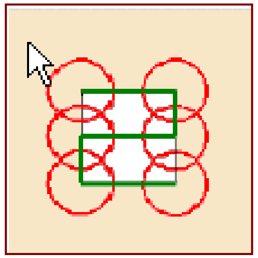
Overhang = 50%

Overhang = 0%

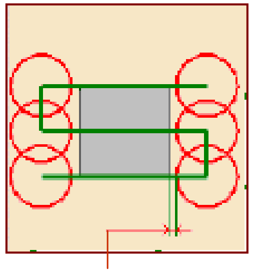


End of Path

IN

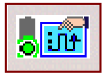


OUT



Tool Side Approach Clearance

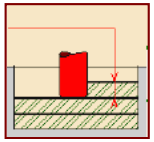
Facing Operation: Strategy (4/5)

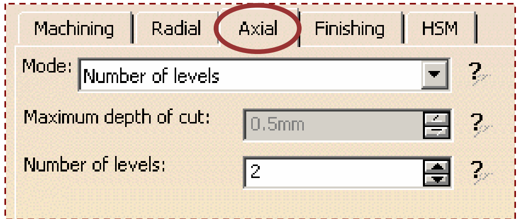


Axial Strategy Parameters:

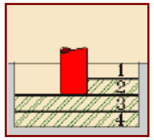
Axial Strategy Mode:

Maximum Depth of Cut: Maximum depth of cut in axial direction between 2 levels

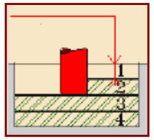




Number of Levels: Number of levels to machine



Number of Levels Without Top: To define the amount of material to cut in the axial direction without Top Plane definition

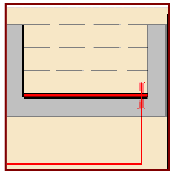


Bottom Finish Pass parameters:

Finish Pass Mode:

No Finish Pass: No special finish pass

At Bottom Only: Indicates that a finish pass is to be generated on the bottom plane of the Facing Operation. A thickness value must be given for that pass

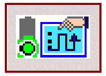


The Finishing Feed rate will be used to cut the material on the finish pass.

Student Notes:

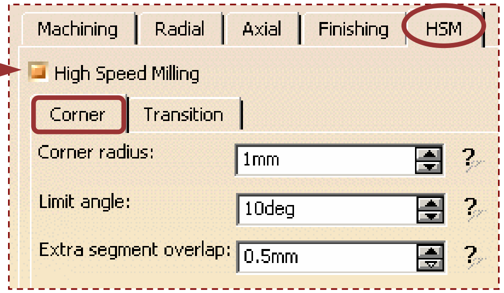
Student Notes:

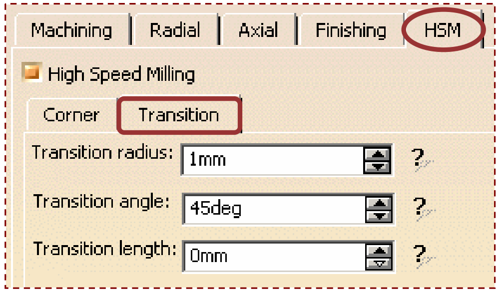
Facing Operation: Strategy (5/5)



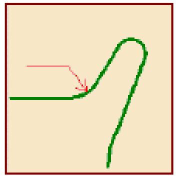
High Speed Milling (HSM) Parameters:
(only in Inward Helical)

This button activates high speed milling

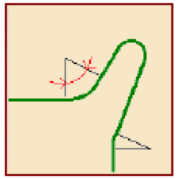




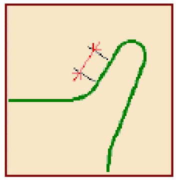
Corner Parameters



Corner radius

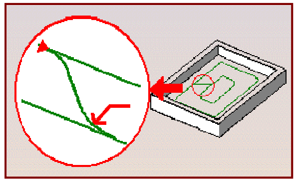


Limit Angle: Lowest angle between two segments that needs cornering

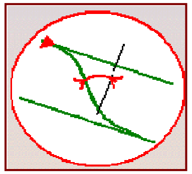


Extra segment overlap

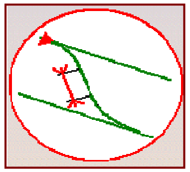
Transition Parameters



Transition radius



Transition angle



Transition length

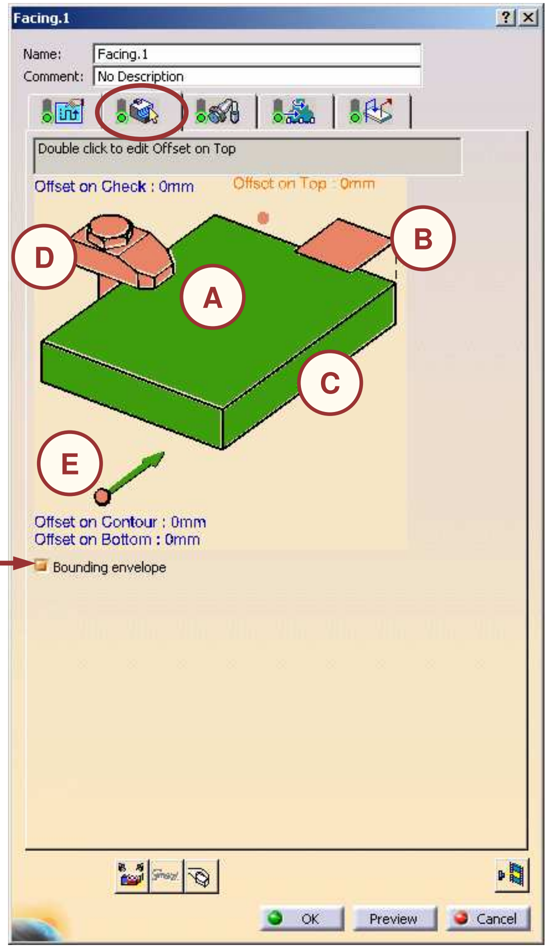
Student Notes:

Facing Operation: Geometry

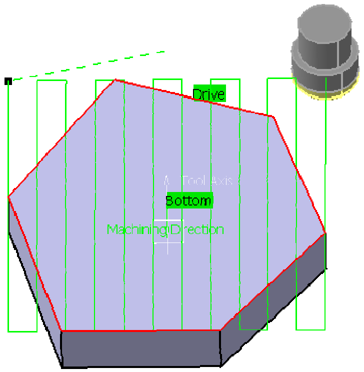
You will learn how to select a Geometry for a Facing Operation.

This Geometry Tab Page includes a Sensitive Icon dialog box that allows the selection of:

- (A) **Bottom Plane**
- (B) **Top Plane (only for Multi-Level operations)**
- (C) **Drive Elements**
- (D) **Check Elements (Optional)**
- (E) **Start Point (Optional)**



Offsets can be applied on the Top Plane, Bottom Plane, Contour and Check Elements (Double-click the value)



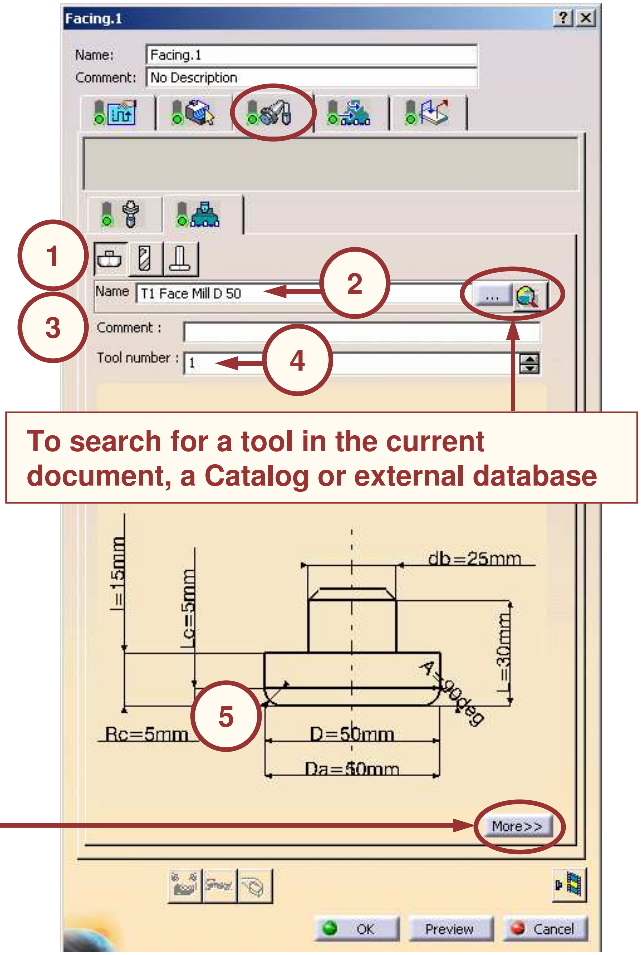
The system automatically computes the bounding rectangle of the part along the Machining Direction in Back & Forth and One Way

Student Notes:

How to Select/Create a Tool

You will learn the general process to select/create a Tool to perform a Facing Operation.

1. **Select the tool type available for the current operation**
2. **Type the Name of the Tool.**
3. **Type a text of comment (Optional)**
4. **Specify a new tool number (already does not exist)**
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 's parameters such as Geometry, Technology, Feeds & Speeds and Compensation.

Student Notes:

General Process to Select Feeds and Speeds

The Selection of Feeds and Speeds has been elaborated in detail as follows:

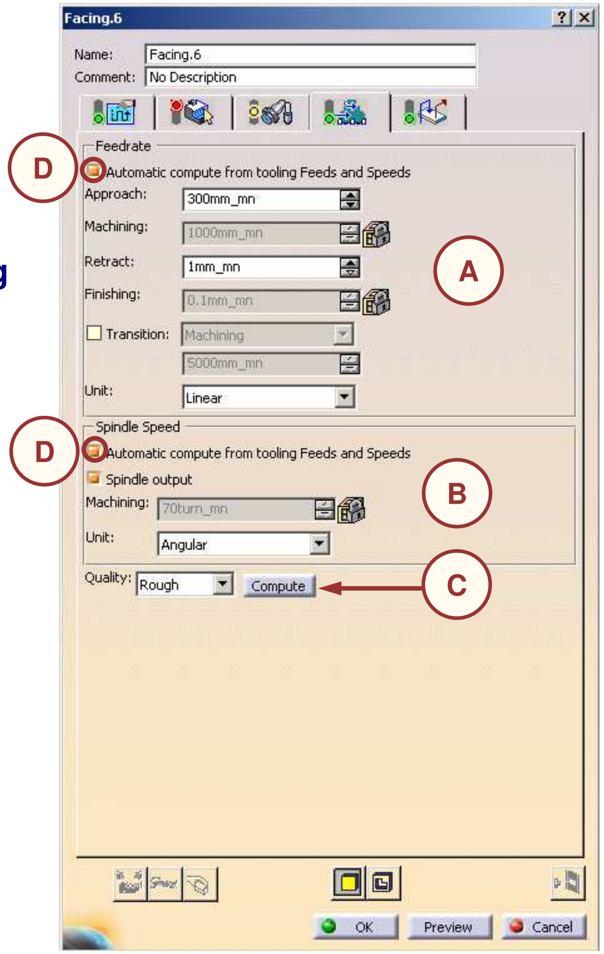
A **Define the Feedrate values (according to the unit Linear [mm/mn] or Angular [mm/turn]) for**

Approach Feedrate: This feedrate is used by default during approach motion

Machining Feedrate: This feedrate is used during Machining motion

Retract Feedrate: This feedrate is used by default during retract motion

Finishing Feedrate: This feedrate is used as Machining Feedrate for the Bottom Finish Pass



B **Define the Spindle Speed value according the unit Linear (m/mn) or Angular (turn/mn)**

You can exclude Spindle speed information from the NC data output by deactivating the check box Spindle Output

C **Select Quality (Rough, Finish, Either) and compute it, according to value defined on the tool**

D **You can select « automatic compute » then feedrate and spindle speed will be automatically compute from tooling Feeds and Speeds**

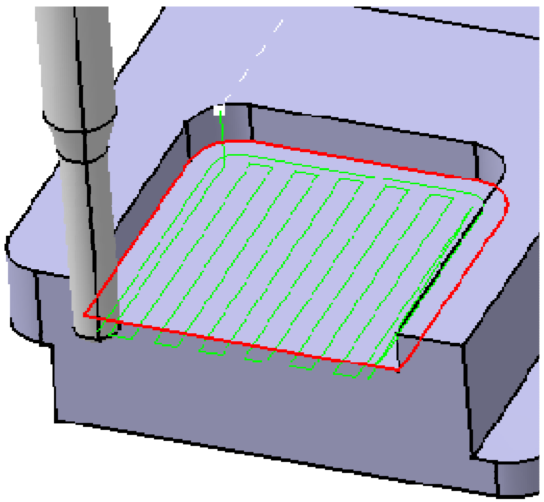
Student Notes:

Pocketing Operation

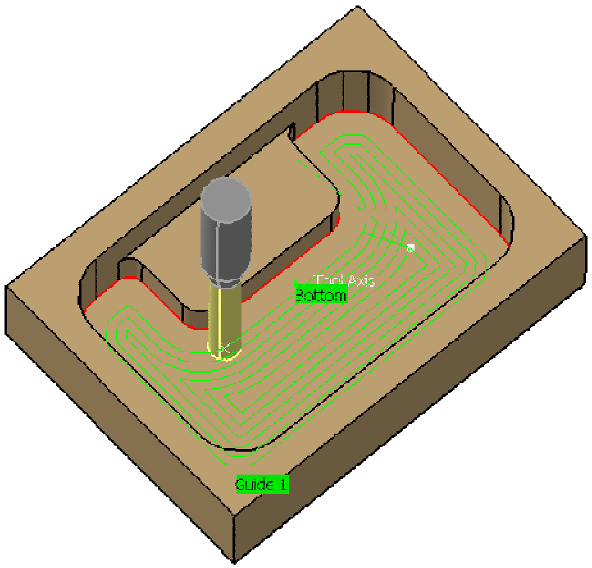
In this lesson, you will learn what is a Pocketing Operation, types of a Pocketing Operation and how to create them.

- Introduction
- Creating a Pocketing Operation
- Strategy
- Geometry
- Feeds and Speeds
- Dedicated Approach Macro

Open Pocket



Closed Pocket



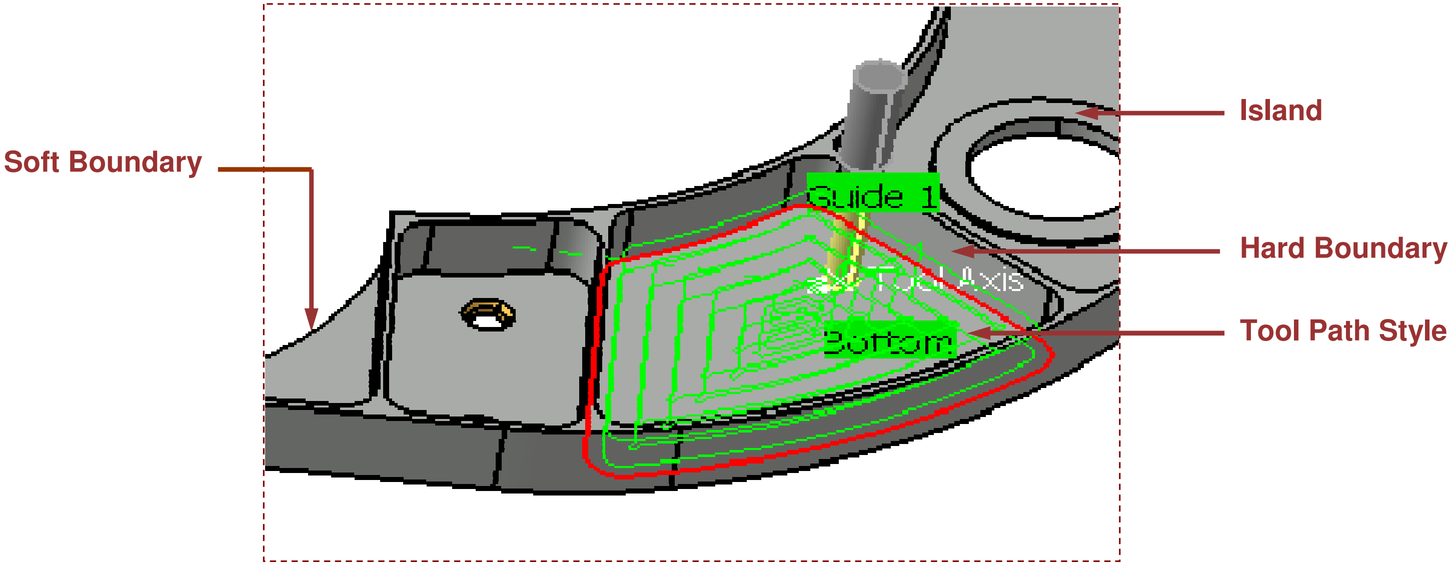
Student Notes:

Pocketing Operation: Introduction

A Pocketing Operation is used to machine a pocket with or without inner domains.

In Pocketing Operation:

- **The external boundary is made of Hard/Soft elements.**
- **The islands are limited only with Hard boundaries.**
- **The material will be removed in one or several levels of cut.**
- **The tool starts and finishes machining at the top of the pocket.**
- **The pocket is machined with Outward and Inward helical or Back & Forth tool path style.**



Student Notes:

Creating a Pocketing Operation: General Process

1. Type the Name of the Operation. (optional because a default name is given by the system 'Type_Of_Operation.X')

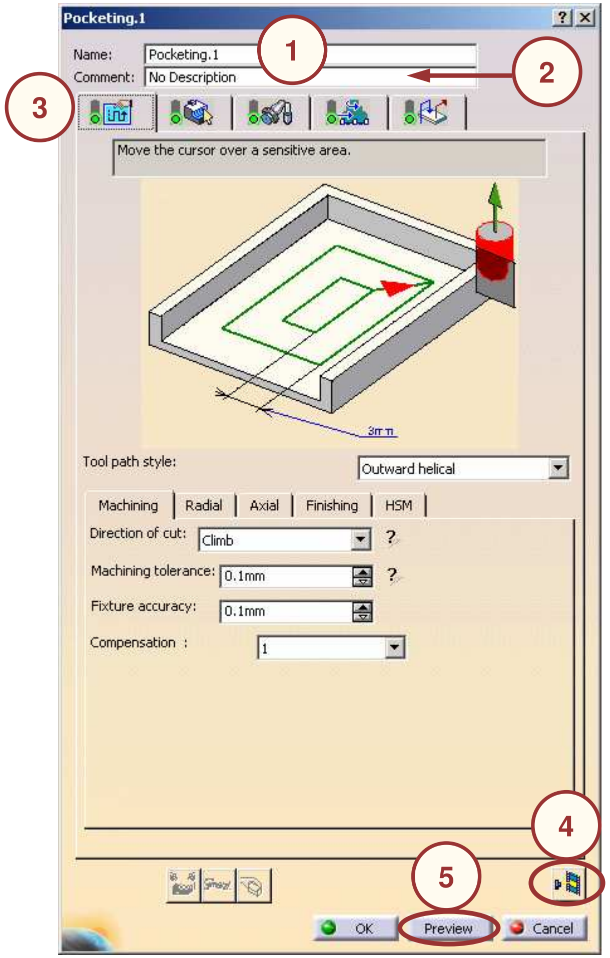
2. Type the text of comment (optional)

3. Define operation parameters using the 5 tab pages

- Strategy tab page
- Geometry tab page
- Tool tab page
- Feeds & Speeds tab page
- Macros tab page

4. Replay and/or Simulate the operation tool path

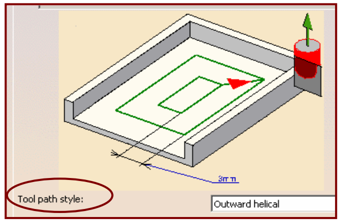
5. Before replaying or creating the operation, "Preview" checks that all parameters are coherent



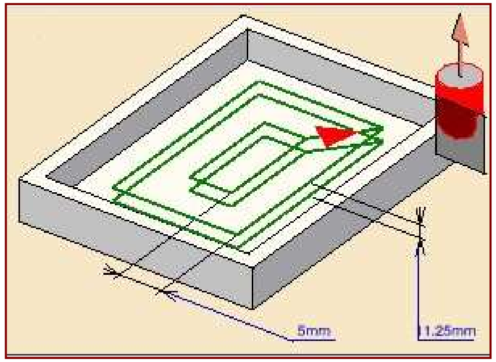
Student Notes:

Pocketing Operation: Strategy (1/9)

The three possible "Tool path styles" for a Pocketing Operation are as shown below:

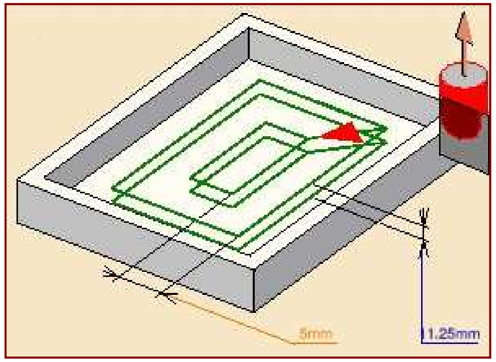


Outward Helical



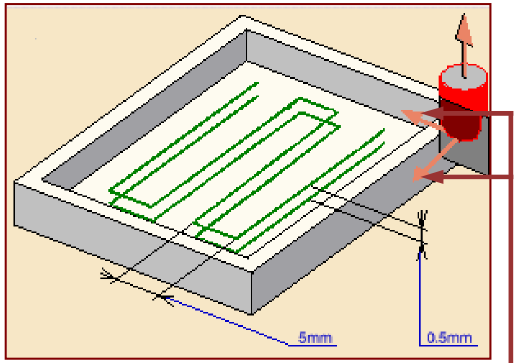
The tool starts from the center and follows outward paths parallel to the boundary avoiding islands

Inward Helical



The tool starts from the contour and follows inward paths parallel to the boundary avoiding islands

Back & Forth

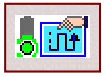


The tool alternatively machines in one direction then in opposite direction and follows the machining direction

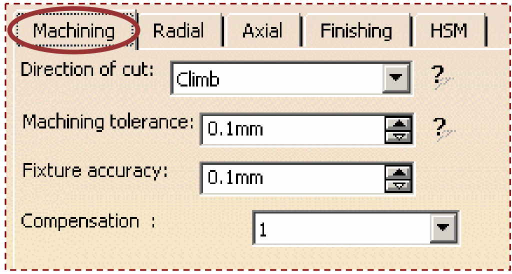
To change the machining or progression direction select either arrow

Student Notes:

Pocketing Operation: Strategy (2/9)



Machining Strategy Parameters:

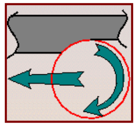


Direction of Cut

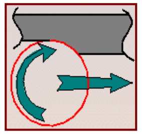
Climb: The front of the advancing tool cuts into the material first

Conventional: The back of the advancing tool cuts into material first

Climb Milling

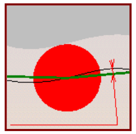


Conventional Milling



Machining Tolerance

Value of the maximum allowable distance between theoretical tool path and the computed tool path



Fixture Accuracy

Local machining tolerance for fixtures

Compensation

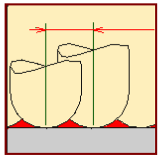
Number of the tool compensation used (if this is already defined on the tool).

Student Notes:

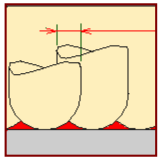
Pocketing Operation: Strategy (3/9)

Radial Strategy Parameters:

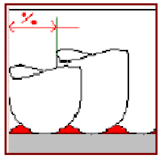
Maximum Distance
A maximum distance will be used to compute the distance between two paths.

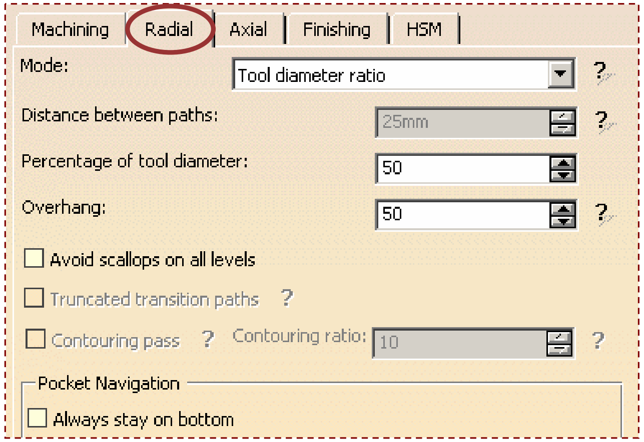


Tool Diameter Ratio
The distance between two paths will respect a tool diameter overlap ratio.



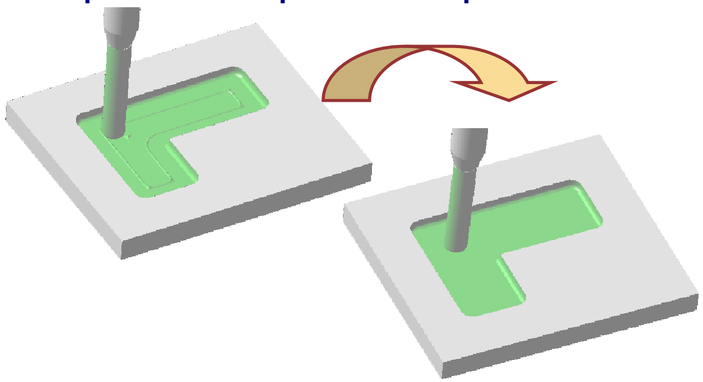
Stepover Ratio
The distance between two paths with respect to a stepover ratio.
(10% stepover = 90% tool diameter)





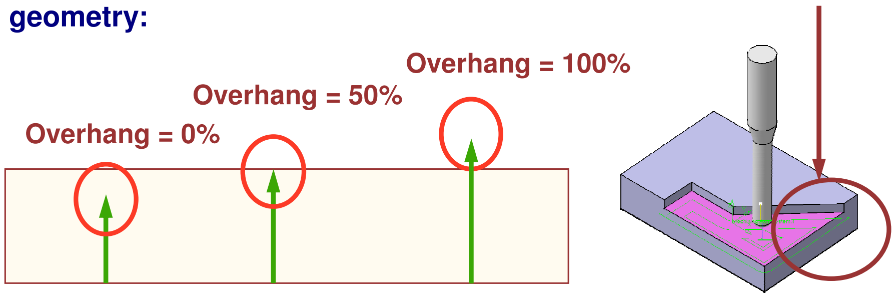
Avoid scallops on all levels:

In all the machining levels (including bottom finish level if defined) the tool path is computed with respect to scallop avoidance



Overhang: Extension of the tool path in percentage of the tool diameter. Only available for open pockets.

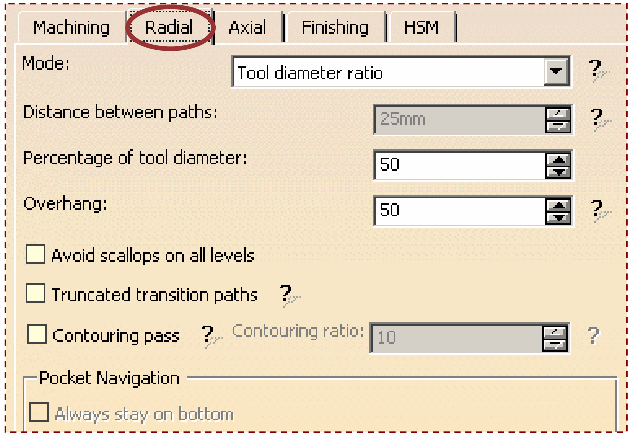
Overhang of 100% is useful to machine properly this kind of geometry:



Student Notes:

Pocketing Operation: Strategy (4/9)

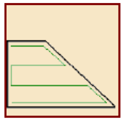
Radial Strategy Parameters:

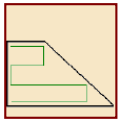


For Back & Forth strategy only:

Truncated transition paths

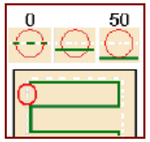
Enables the tool to follow the external profile more exactly by allowing the transition portion of the trajectory to be truncated.





Scallop pass

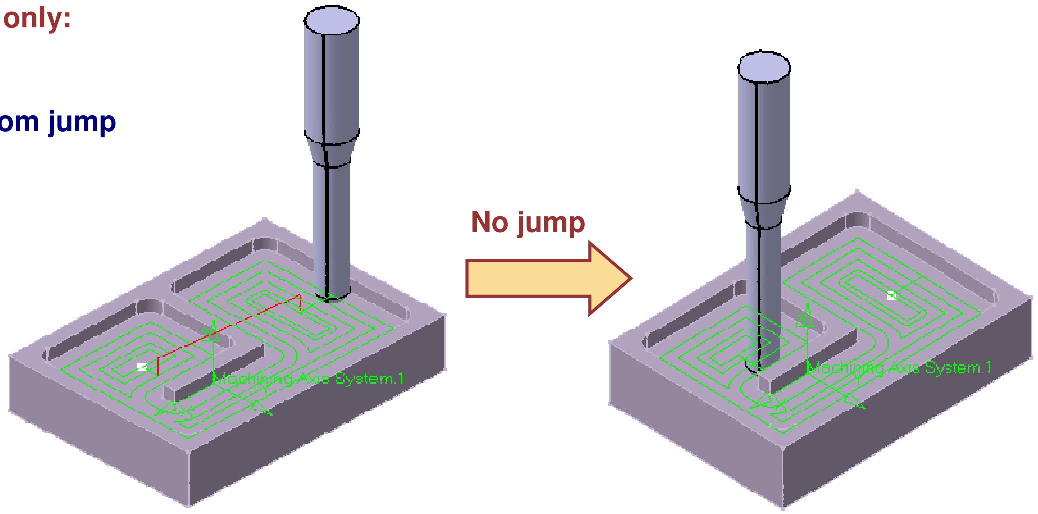
Allow to remove scallop on contour driven by a ratio of tool diameter



For Inward and Outward helical strategy only:

Pocket Navigation

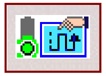
Always stay on bottom option prevent from jump between different area during machining



'Always stay on bottom' deactivated **'Always stay on bottom' activated**

Student Notes:

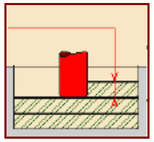
Pocketing Operation: Strategy (5/9)

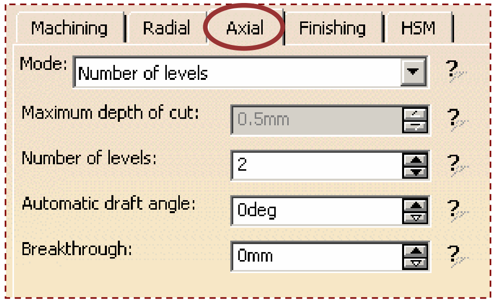


Axial Strategy Parameters:

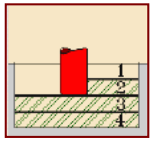
Axial Strategy Mode:

Maximum Depth of Cut:
Maximum depth of cut in axial direction between two levels

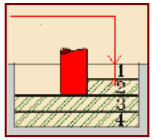




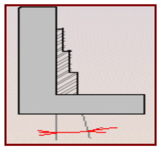
Number of Levels:
Specify the number of levels from the bottom to the top



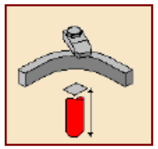
Number of Levels Without Top: To define the amount of material to be cut in the axial direction without Top Plane definition, Specify the bottom, the number of levels and the depth of cut.



Automatic draft angle
Incremental increase of thickness on flank

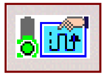


Breakthrough
Only in soft bottom pocket, You give an offset in order to specify a virtual bottom.



Student Notes:

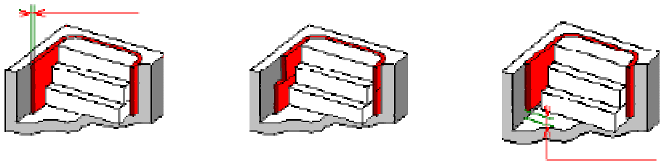
Pocketing Operation: Strategy (6/9)

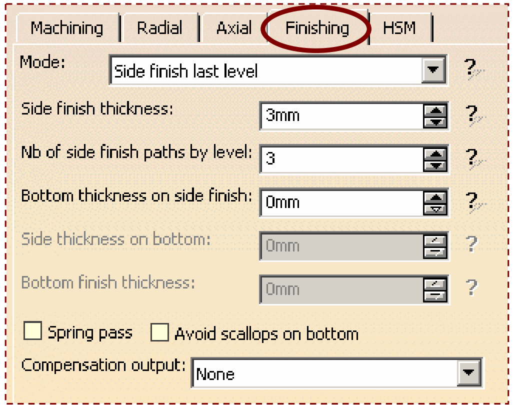


Finishing Strategy Parameters:

Side finish last level

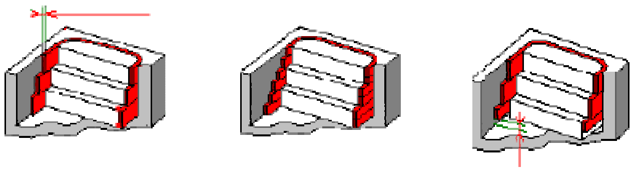
Activate a radial finish pass only at the last level of the operation. Specify Side finish thickness, Number of paths by levels and a bottom thickness to respect.





Side finish each level

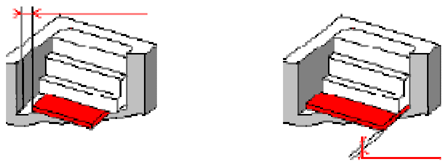
Activate a radial finish pass at each level of the pocketing. Specify Side finish thickness, Number of paths by levels and a bottom thickness to respect.



The Finishing Feedrate will be used to cut the material on the finish pass.

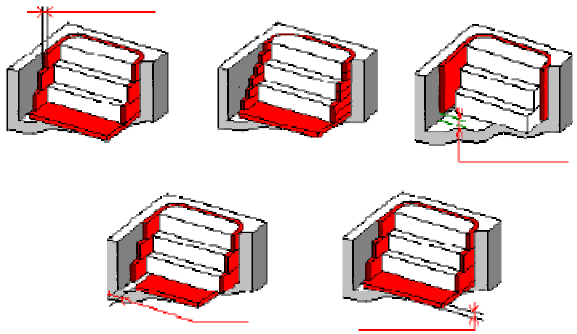
Finish bottom only

Activate a last level pass to machine the bottom. Specify a Side thickness on bottom to respect and a Bottom finish thickness.



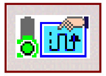
Side finish at each (or last only) level & bottom

Activate one or many radial finish passes at each level (or last level only) and a last level pass to machine the bottom



Student Notes:

Pocketing Operation: Strategy (7/9)

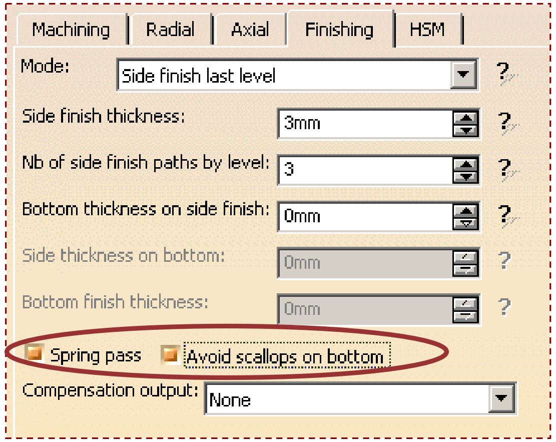


Finishing Strategy Parameters:

Special Finish Pass

Spring Pass
Duplicate last finish pass to compensate the spring of the tool

Avoid scallops on bottom
Modify distance between pass on last level to avoid scallop on bottom.
This option is not available if the option 'Avoid scallops on all levels' has already been activated



Student Notes:

Pocketing Operation: Strategy (8/9)

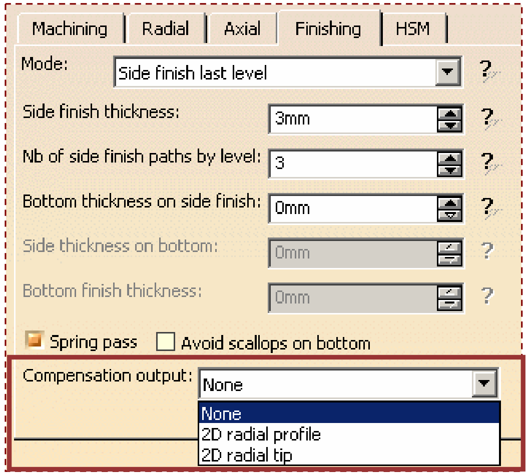
Finishing Strategy Parameters:

Compensation output
Generation of Cutter Compensation (CUTCOM) instructions for the pocketing side finish pass

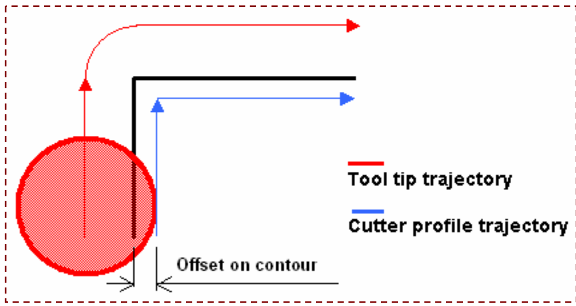
- **None**
  Cutter compensation instructions are not generated in the NC data output (one can define them manually)

- **2D radial profile**
  Both the tool tip and cutter profile will be visualized during tool path replay. Cutter compensation instructions are automatically generated in the NC data output based on cutter profile trajectory. An approach macro must be defined to allow the compensation to be applied.

- **2D radial tip**
  Tool tip will be visualized during tool path replay. Cutter compensation instructions are automatically generated in the NC data output based on tool tip trajectory. An approach macro must be defined to allow the compensation to be applied.



A negative Offset on contour (parameter in Geometry tab page) is possible for 2D radial profile



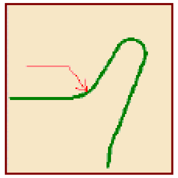
Any user-defined PP words in macros are added to the cutter compensation instruction generated in the NC data output. Therefore be careful when specifying CUTCOM instruction in macros!

Student Notes:

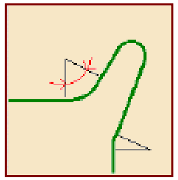
Pocketing Operation: Strategy (9/9)

High Speed Milling Strategy Parameters:

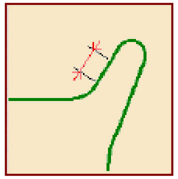
Corner Parameters:



Corner radius

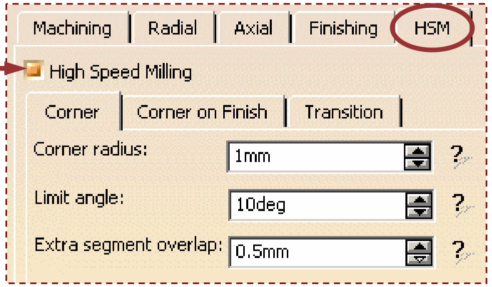


Limit Angle: Lowest angle between two segments that needs cornering



Extra segment overlap

This button activates High Speed Milling Strategy

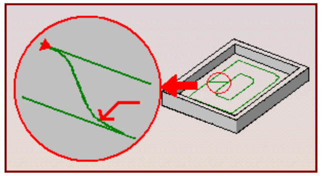


Corner on Finish Pass Parameters:

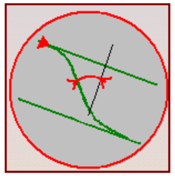
Corner radius: Radius applied to cornerize the Side finish pass

Limit Angle: Minimum angle between two segments that needs cornerization on the Side finish pass

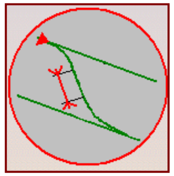
Transition Parameters (only for Inward or Outward Helical)



Transition radius



Transition angle



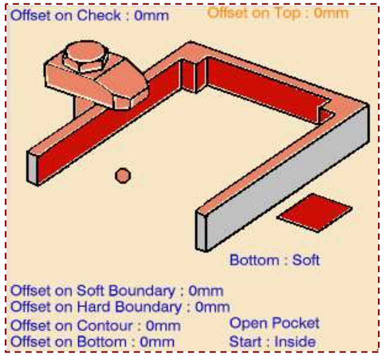
Transition length

<u>Student Notes:</u>

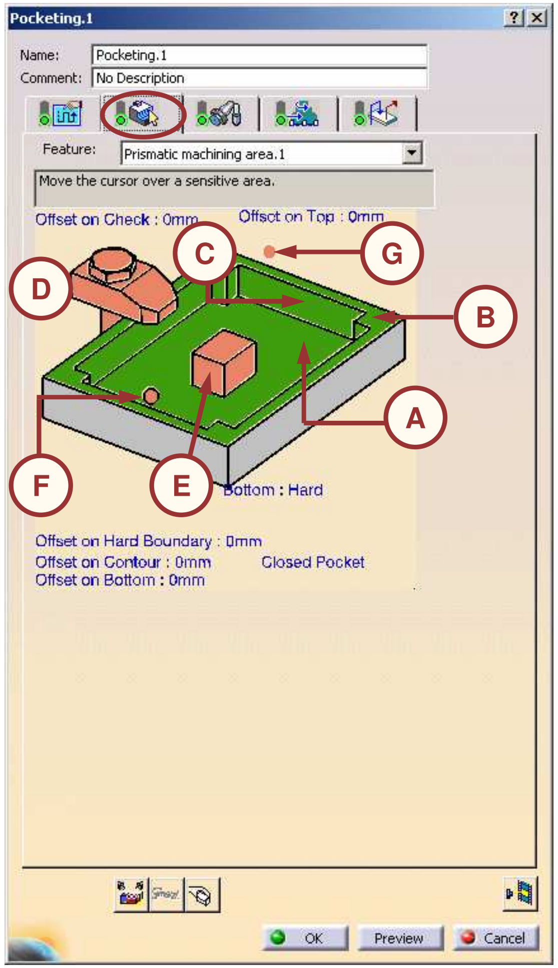
Pocketing Operation: Geometry (1/4)

This Geometry Tab Page includes a Sensitive Icon dialog box that allows the selection of:

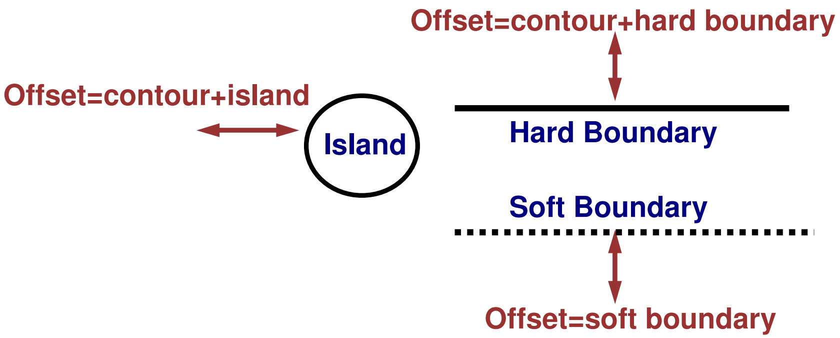
- **A** Bottom Plane
- **B** Top Plane (only in Multi-Levels strategy case)
- **C** Drive Elements
- **D** Check Elements
- **E** Islands
- **F** Start Point
- **G** End Point



To remove the bottom click on: Bottom : Hard



Using contextual menu, offsets can be applied on the Top Plane, Bottom Plane, Contour, Islands, Check Elements, Soft and Hard Boundaries

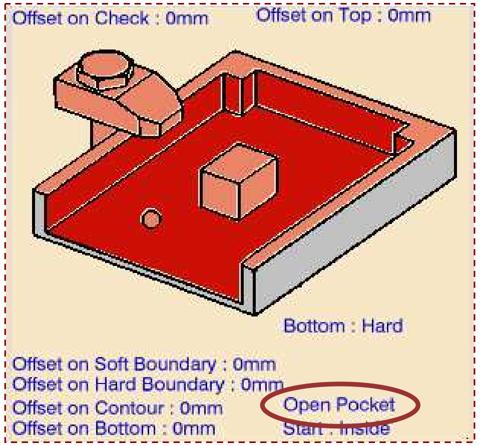


Pocketing Operation: Geometry (2/4)

To machine an open pocket click: Close pocket

Soft boundaries will be automatically detected when selecting bottom (dotted lines)

If you need to add more soft boundaries, select them after bottom selection



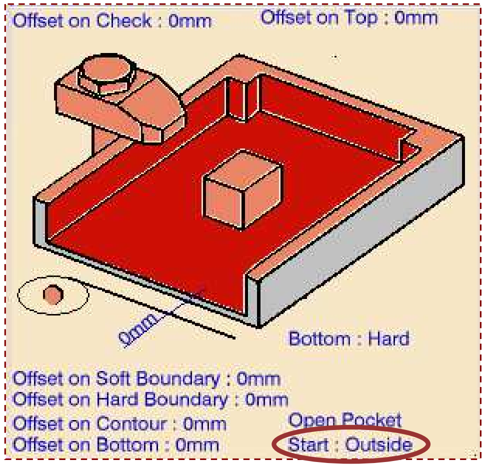
To allow Start point definition outside the machining domain click: Start : Inside

In this case you can specify:

- **A clearance**

  or
- **select an edge and give a clearance**

  or
- **select a point**



Student Notes:

Student Notes:

Pocketing Operation: Geometry (3/4)

You can select a Start Point (1) and an End Point (2) as preferential start and end positions

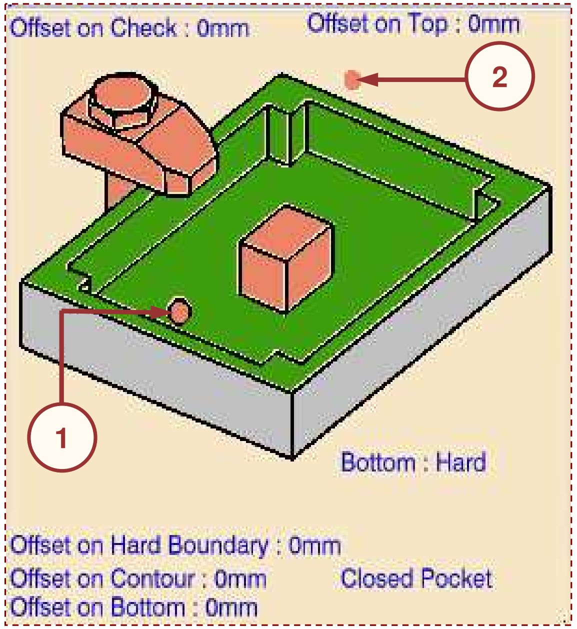
Note that you can select a point or an edge as Start/End Point in Outward/Inward strategy.

In case of selecting an edge, the point will be chosen near the middle of this edge (according to the selected geometry, overhang, offset)

Start point will always be respected except if a ramping motion is defined in the macro (the start point will be near the selection)

Note that End point is only available in Outward Strategy and it might be modified (according to the selected geometry and computed tool path)

Note that you can specify a Start Point outside the machining domain for an open pocket (see previous slide)



Student Notes:

Pocketing Operation: Geometry (4/4)

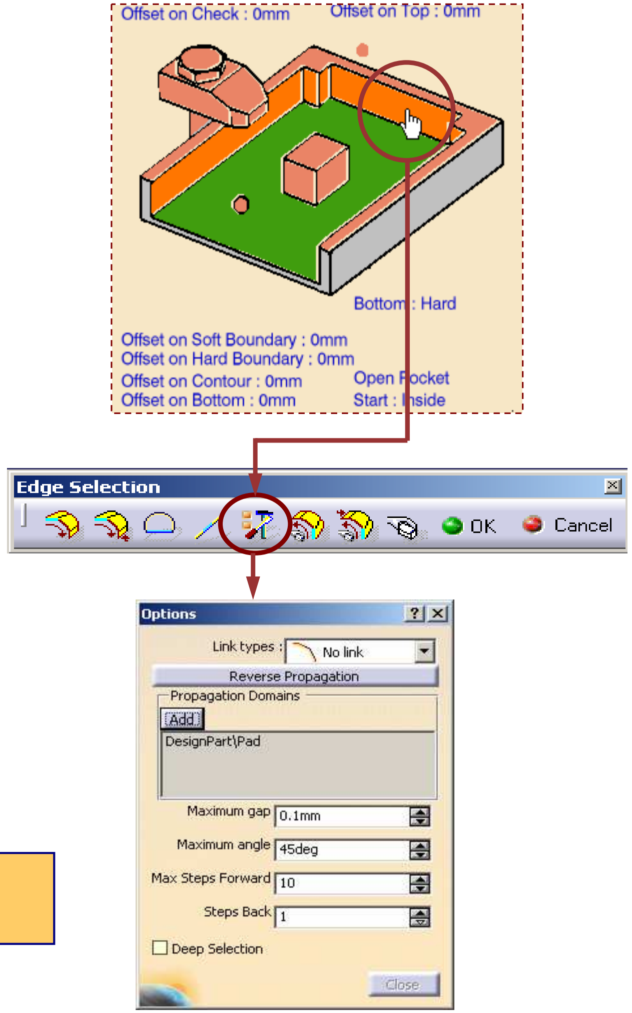
Guiding element:
Edge Selection wizard

Adds an option to manage the contour creation during the selection of elements:

- **No link**
- **Automatic link**
- **Line Insert**
- **Linear extrapolation**

Manage the connection of the guiding elements:

connect contour to another, connect all contours, delete connecting element (line or extrapolated lines) between elements



Selected elements are kept even if the contour is temporary opened during pocketing selection.

Student Notes:

Pocketing Operation: Feeds and Speeds (1/2)

Feedrate Reduction in Corners:

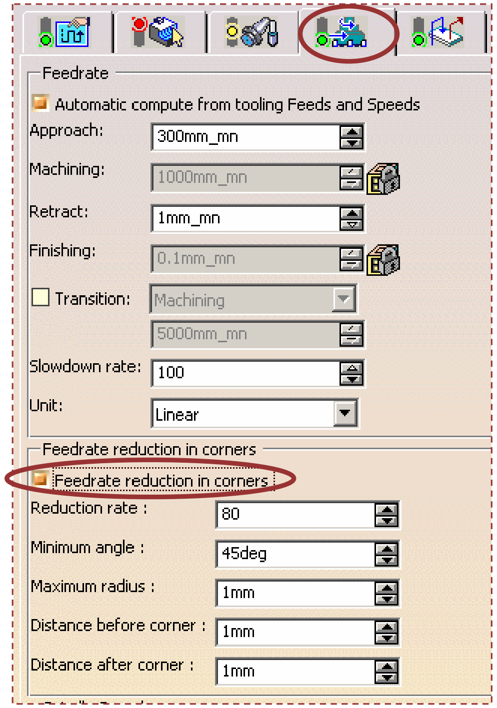
You can reduce feedrates in corners encountered along the tool path depending on the values given in the Feeds and Speeds tab page

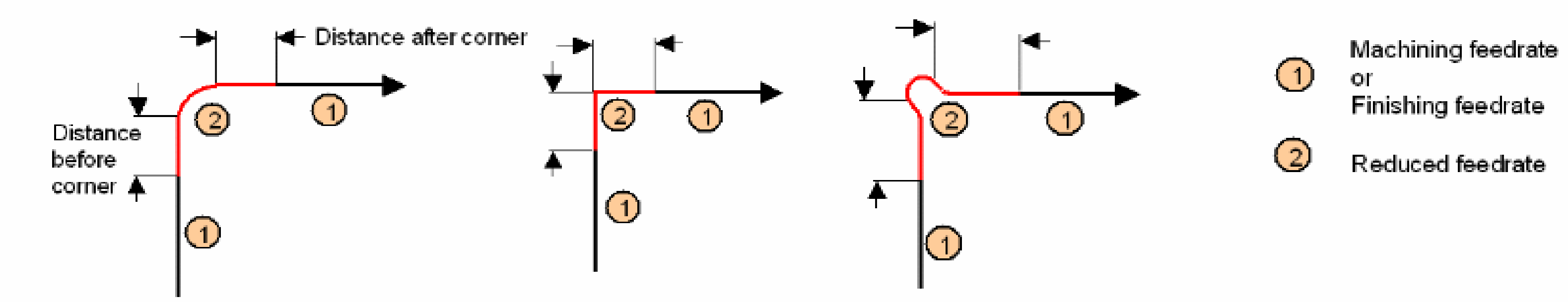
Feed reduction is applied to the corners along the tool path whose radius is less than the Maximum radius value and whose arc angle is greater than the Minimum angle value.

For Pocketing, feedrate reduction applies to inside and outside corners for machining or finishing passes.

It does not apply for macros or default linking and return motions.

Corners can be angled or rounded, and may include extra segments for HSM operations.



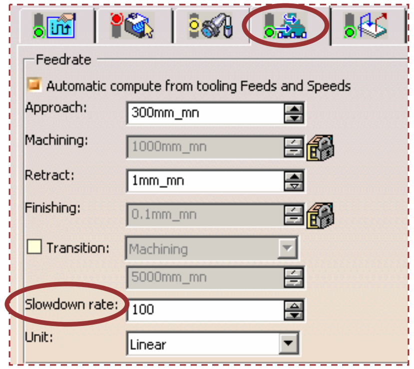


Student Notes:

Pocketing Operation: Feeds and Speeds (2/2)

Slowdown Rate Option:

You can use Slowdown Rate in the Feeds and Speeds tab page to reduce the current feedrate by a given percentage.



In Outward/Inward Helical Pocketing, the reduction is applied to the first channel cut.

In Back and Forth Pocketing, the reduction is applied to the first channel cut and to the transitions between passes.

Combining Slowdown Rate and Feedrate Reduction in Corners

If a corner is included in a Slowdown path, the general rule is that the lowest percentage value is taken into account.

For example, if the Slowdown rate is set to 70 % and Feedrate reduction rate in corners is set to 50%, the feedrate sequence is:
100%, 70% (entry in slowdown), 50% (entry in corner), 70% (end of corner, still in slowdown), 100% (end of slowdown).

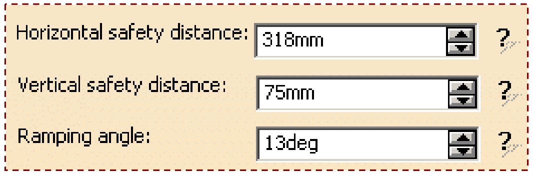
If Feedrate reduction rate in corners is then set to 75%, the feedrate sequence is:
100%, 70% (entry in slowdown), 70% (entry in corner: 75% ignored), 70% (end of corner, still in slowdown), 100% (end of slowdown).

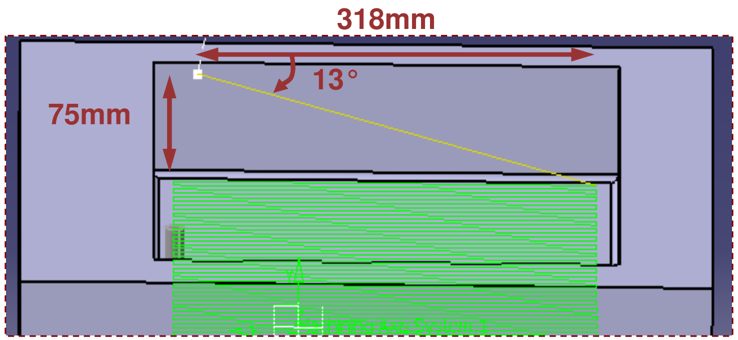
Student Notes:

Pocketing Operation: Dedicated Approach Macro (1/2)

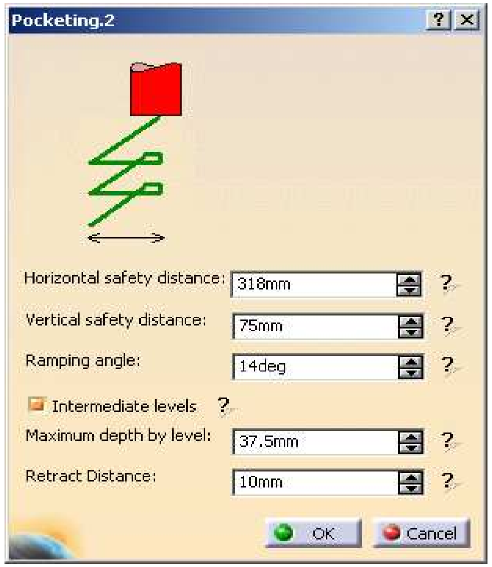
Ramping Approach Macro:

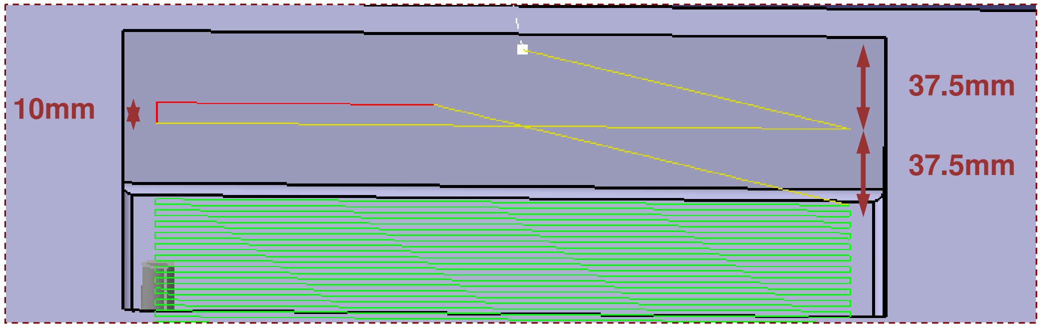
Regular Ramping Approach Macro:





Pocketing Dedicated Ramping Approach macro:



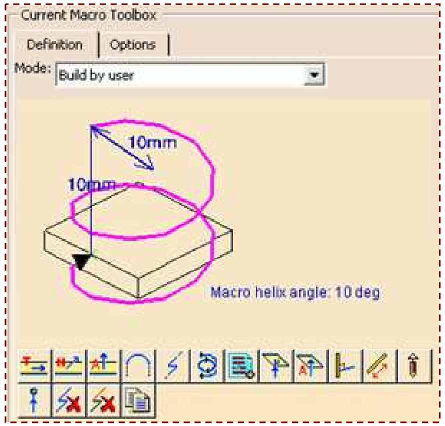


Student Notes:

Pocketing Operation: Dedicated Approach Macro (2/2)

Helix Approach Macro: 

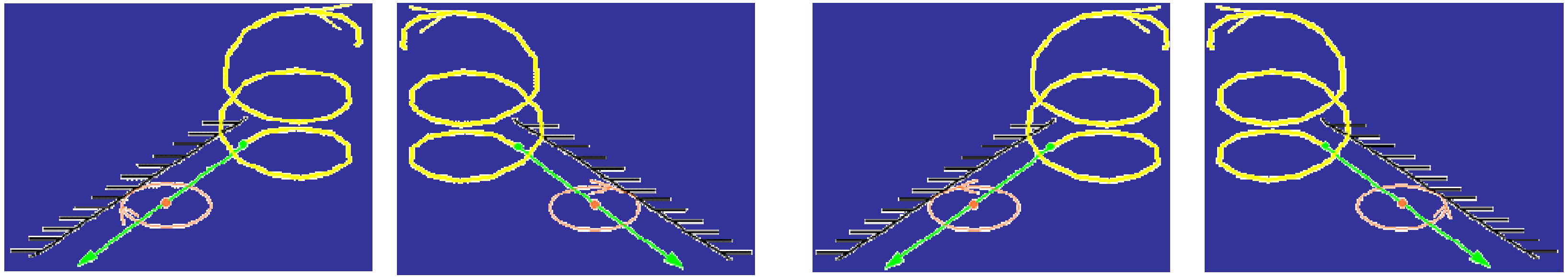
To improve the approach motion in Pocketing operation, you can define Helix Approach Macro. Using the helix macro, the cutter will approach the raw material in helix motion.



This helix macro is available in Build by user mode.
You can modify the helix by double-clicking on pink helix and changing its radius, height and angle values.

Illustrated below the cases of helix macro in relation with 'Direction of cut' and 'Way of rotation of tool':

Way of rotation of tool: Right | **Way of rotation of tool: Left**



Direction of cut: Climb | **Direction of cut: Conventional** | **Direction of cut: Climb**

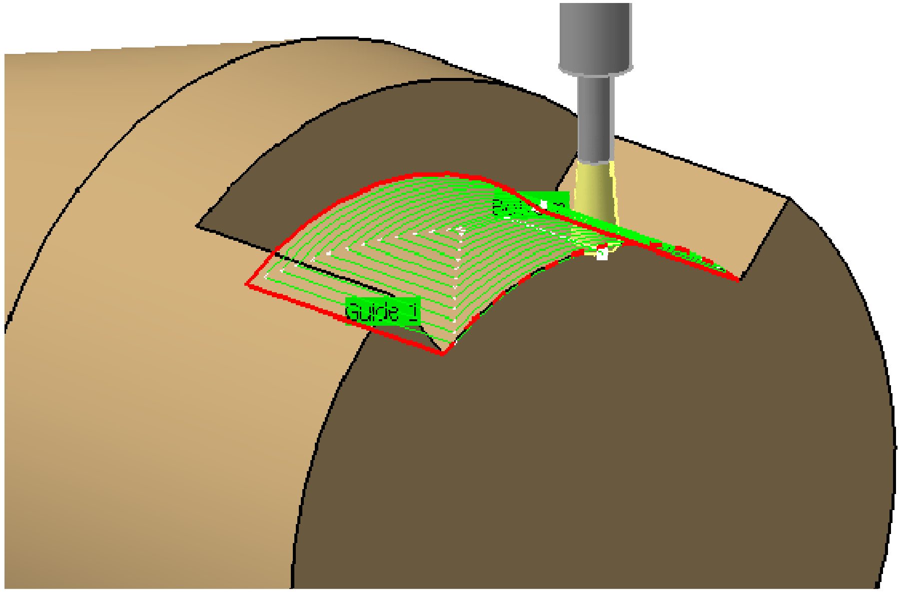
The helix approach macro is available only for Pocketing and Profile Contouring in PMG.

Student Notes:

4-Axis Pocketing Operation

In this lesson, you will become familiar with 4-Axis Pocketing Operation.

- Introduction
- Creating a 4-Axis Pocketing Operation
- Strategy
- Geometry
- Macro



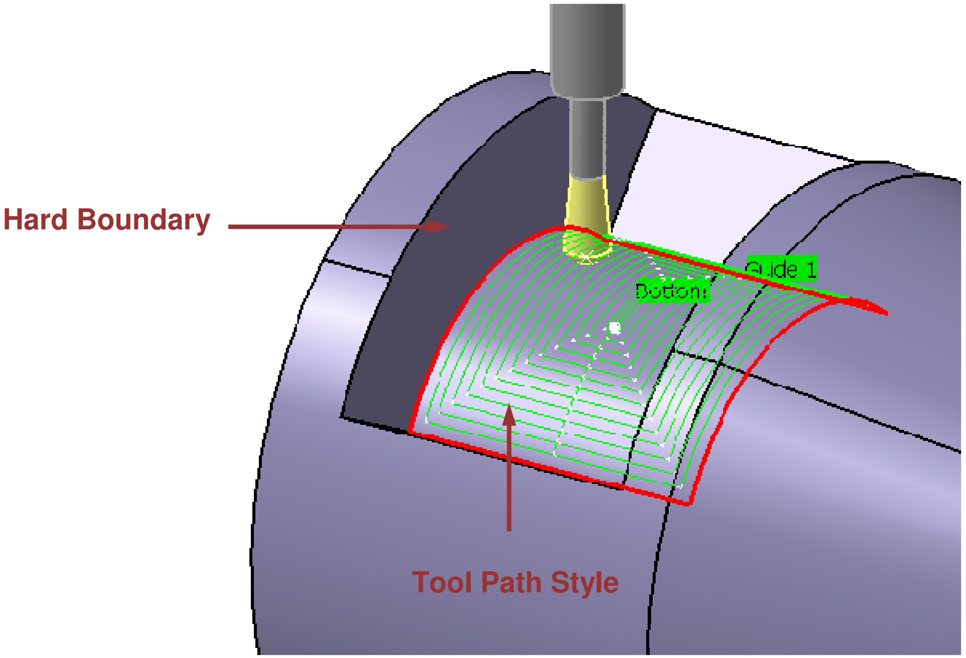
Student Notes:

4-Axis Pocketing Operation: Introduction

A 4- Axis Pocketing Operation is used to machine a pocket on cylindrical or conical surfaces.

In 4-Axis Pocketing Operation:

- **The external boundary is made of Hard/Soft elements.**
- **The material will be removed in one or several levels of cut.**
- **The tool starts and finishes machining from the top of the pocket.**
- **The pocket can be machined with Outward and Inward helical or Back & Forth tool path style.**



Student Notes:

Creating a 4-Axis Pocketing Operation: General Process

1. Type the Name of the Operation.
(optional because a default name is given
by the system 'Type_Of_Operation.X')

2. Type the text of comment (optional)

3. Define operation parameters using the 5 tab pages

Strategy tab page

Geometry tab page

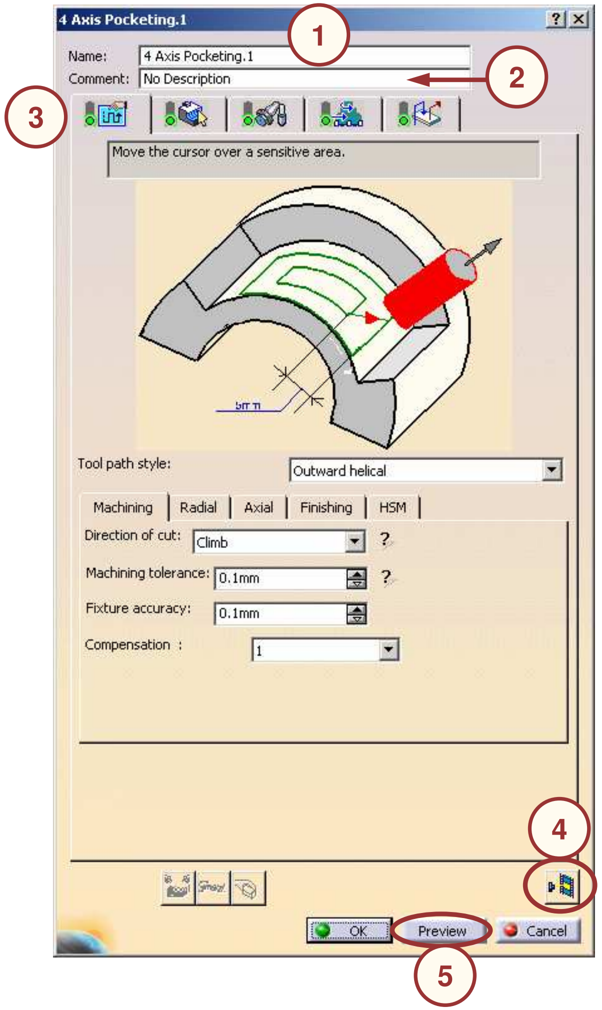
Tool tab page

Feeds & Speeds tab page

Macros tab page

4. Replay and/or Simulate the operation tool path

5. Before replaying or creating the operation, "Preview"
checks that all parameters are coherent



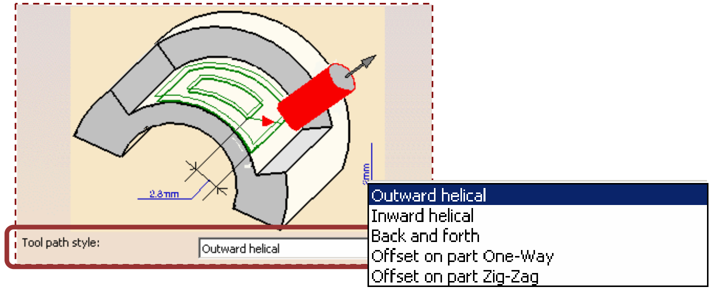
Student Notes:

4-Axis Pocketing Operation: Strategy (1/7)

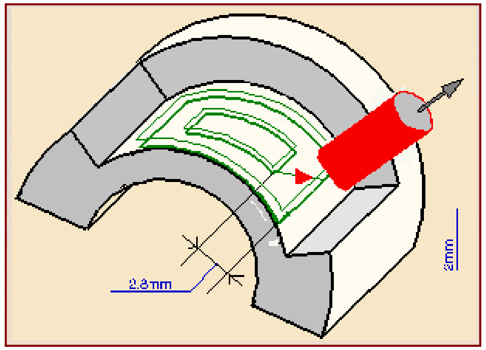
Tool path style:

The Tool path styles decide the cutting modes of the 4-Axis Pocketing operation.

The possible "Tool path styles" for a 4-Axis Pocketing Operation are as shown below:

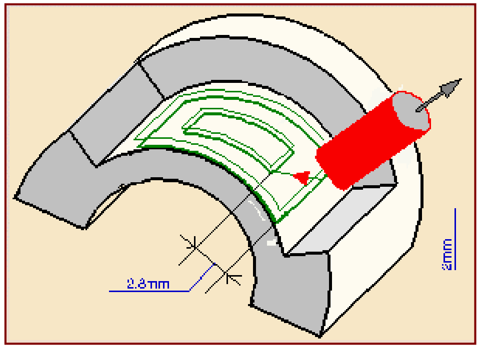


Outward Helical



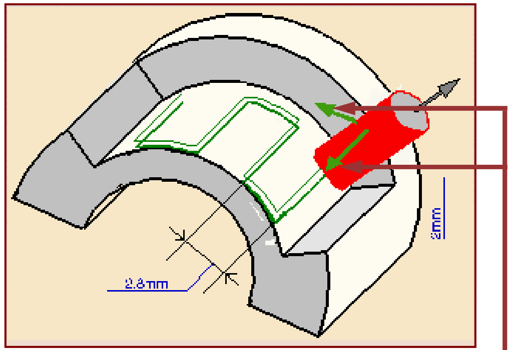
The tool starts from a point inside the pocket and follows outward paths parallel to the boundary.

Inward Helical



The tool starts from a point inside the pocket and follows inward paths parallel to the boundary.

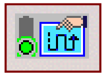
Back & Forth



The tool alternatively machines in one direction then in opposite direction.

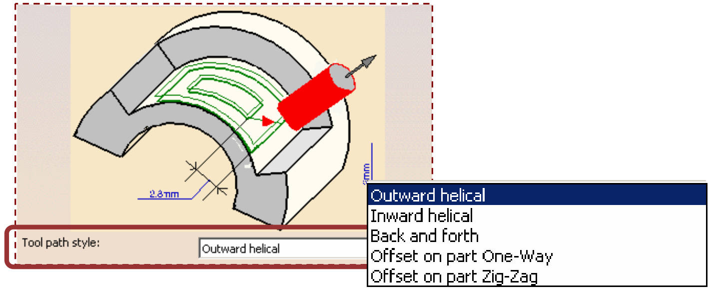
To change the machining or progression direction select either arrow

4-Axis Pocketing Operation: Strategy (2/7)

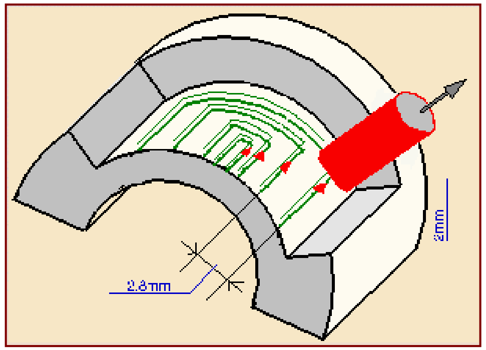


Student Notes:

Tool path style:

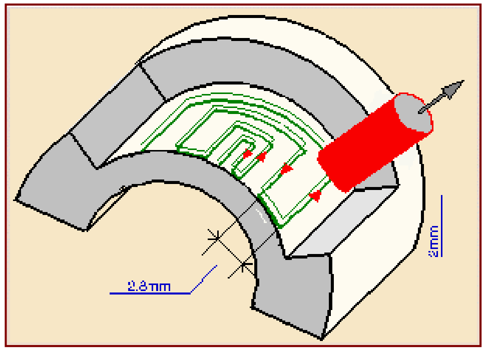


Offset on part One-Way



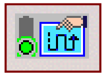
The tool motion is always done in the same direction following paths parallel to the boundary.

Offset on part Zig-Zag



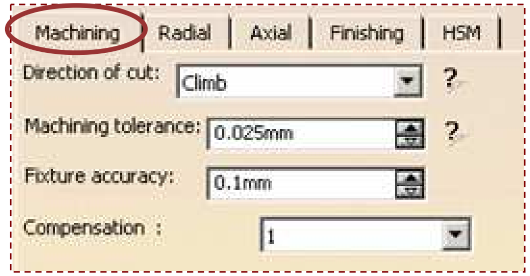
The tool motion is done alternately in one direction and then the other, following paths parallel to the boundary.

4-Axis Pocketing Operation: Strategy (3/7)



Student Notes:

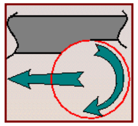
Machining Parameters:



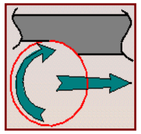
Direction of Cut

Climb: The front of the advancing tool cuts into the material first

Conventional: The back of the advancing tool cuts into material first



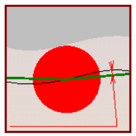
Climb Milling



Conventional Milling

Machining Tolerance

Value of the maximum allowable distance between theoretical tool path and the computed tool path



Fixture Accuracy

Local machining tolerance for fixture thickness.

Compensation

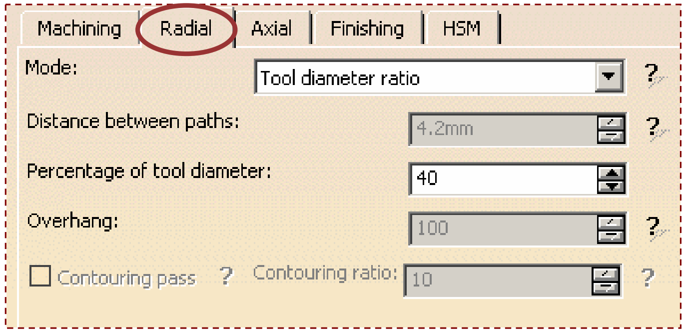
Number of the tool compensation used (if this is already defined on the tool).

Student Notes:

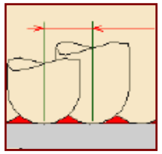
4-Axis Pocketing Operation: Strategy (4/7)

Radial Parameters:

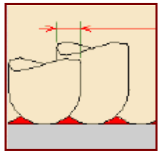
Radial Strategy Mode
It calculates the distance between two consecutive paths of a tool path. There are 3 radial modes:



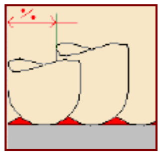
Maximum Distance
A maximum distance will be used to compute the distance between two paths.



Tool Diameter Ratio
The distance between two paths will respect a tool diameter overlap ratio.

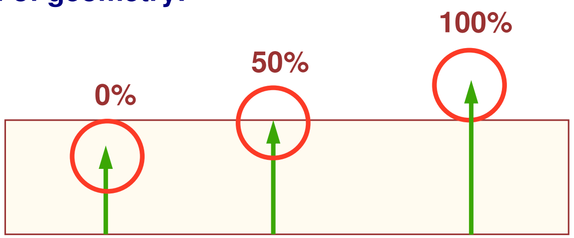


Stepover Ratio
The distance between two paths with respect to a stepover ratio.
(10% stepover = 90% tool diameter)



Overhang: Extension of the tool path in percentage of the tool diameter. Only available for open pockets.

Overhang of 100% is useful to machine properly this kind of geometry:

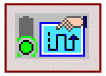


Contouring pass
It is the final machining pass around the exterior trajectory. This option is available for 'Back & Forth' Tool path style and used to remove scallop.

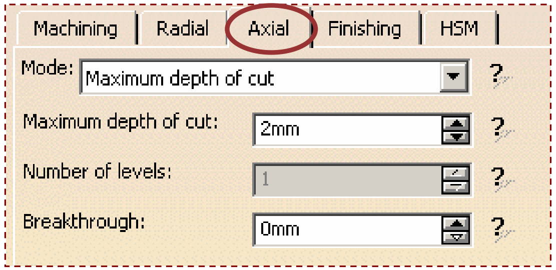
Contouring ratio
It adjusts the position of the final contouring pass by entering a percentage of the tool diameter (0 to 50).

Student Notes:

4-Axis Pocketing Operation: Strategy (5/7)



Axial Parameters:

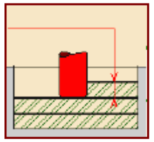


Axial Strategy Mode

It calculates the distance between two consecutive levels of a tool path. There are 3 axial modes:

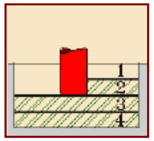
Maximum Depth of Cut:

Maximum depth of cut in axial direction between two levels



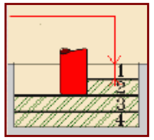
Number of Levels:

The number of levels from the bottom to the top



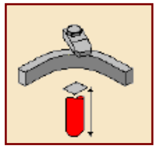
Number of Levels Without Top:

To define the amount of material to be cut in the axial direction without Top Plane definition, specify the bottom, the number of levels and the depth of cut.



Breakthrough

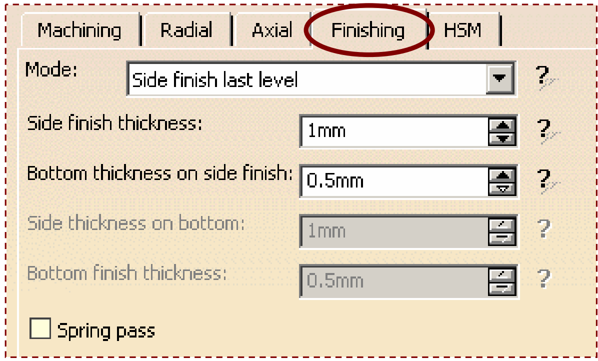
Only in soft bottom pocket, you can give an offset in order to specify a virtual bottom.



Student Notes:

4-Axis Pocketing Operation: Strategy (6/7)

Finishing Parameters:



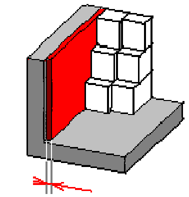
Finishing Strategy Mode

It gives the options whether finish passes are required to be generated on sides and bottom of the area to machine.
Side finishing can be done at each level or only at the last level of the operation.
Bottom finishing can be done without any side finishing or with different combinations of side finishing.

The Finishing Feedrate will be used to cut the material on the finish pass.

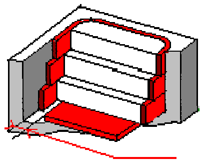
Side finish thickness

The thickness of the material will be machined by the side finish pass.



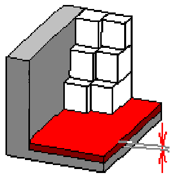
Side thickness on bottom

The thickness of material left on the side by the bottom finish pass.



Bottom finish thickness

The thickness of material that will be machined by the bottom finish pass.



Bottom thickness on side finish

The bottom thickness used for last side finish pass, if side finishing is requested on the operation.

Spring pass

It duplicates the last side finish pass. The spring pass is used to compensate the natural spring of the tool.

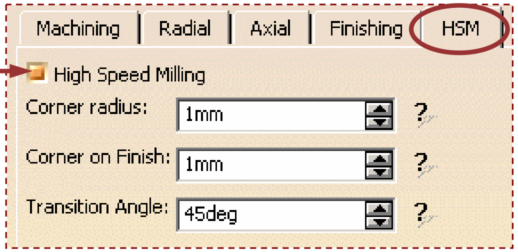
4-Axis Pocketing Operation: Strategy (7/7)

Student Notes:

High Speed Milling Parameters:

These parameters decide whether or not cornering for HSM to be done on the trajectory.

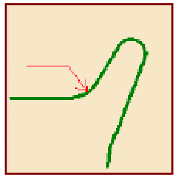
This check box activates High Speed Milling Strategy



Corner Parameters:

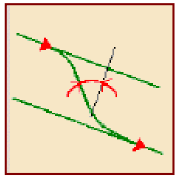
Corner radius: The radius used for rounding the corners along the trajectory of a HSM operation.

Value must be smaller than the tool radius.



Corner on Finish: The radius used for rounding the corners of the side finish path in a HSM operation. Value must be smaller than the tool radius.

Transition Angle: The angle of the transition path that allows the tool to move smoothly from one path to the next in a HSM operation.



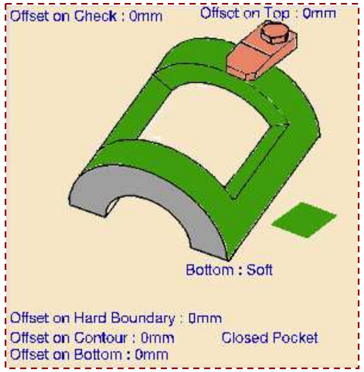
Student Notes:

4-Axis Pocketing Operation: Geometry (1/3)

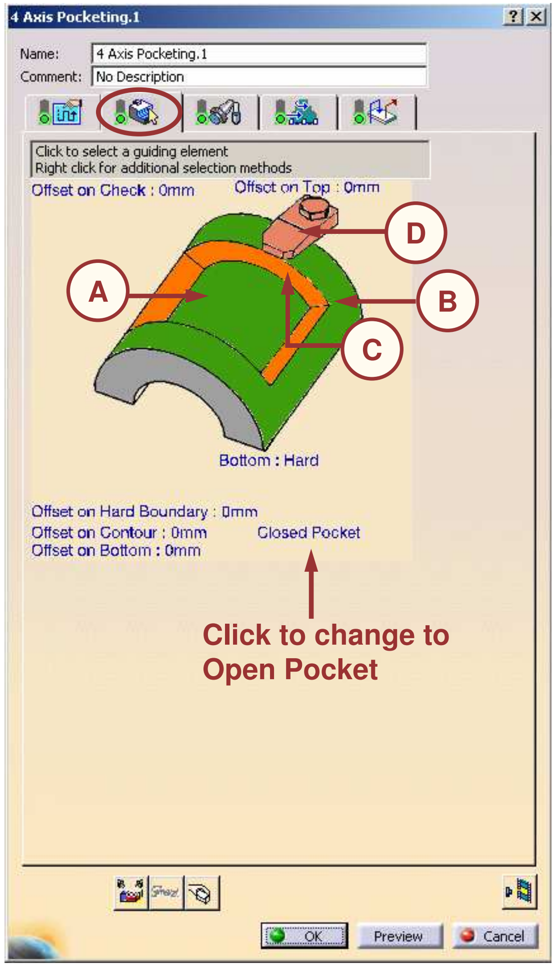
You can create a 4-Axis Pocketing operation for Open Pocket and Closed Pocket.

This Geometry Tab Page includes a sensitive icon dialog box that allows the selection of:

- **A** Pocket Bottom
- **B** Pocket Top (only in Multi-Levels strategy case)
- **C** Pocket Boundary or Guiding element
- **D** Fixture or Check Elements



To remove the bottom click on: Bottom : Hard



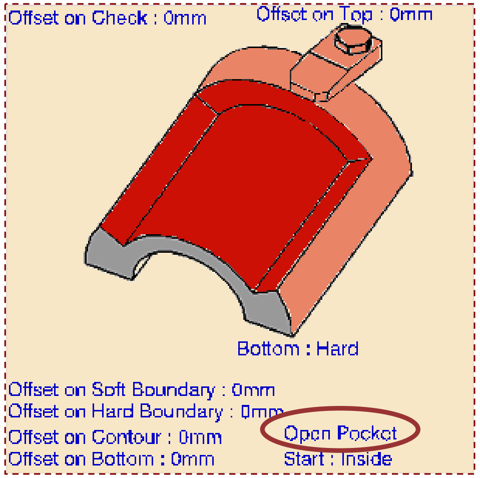
Using contextual menu, offsets can be applied on the Top, Bottom, Contour, Check Elements, and Soft & Hard Boundaries

4-Axis Pocketing Operation: Geometry (2/3)

To machine an open pocket click: Close pocket

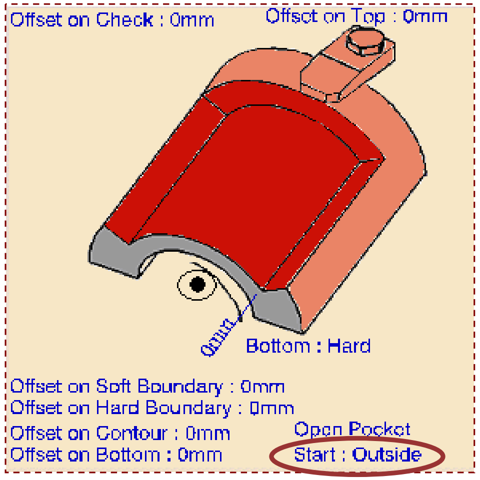
Soft boundaries will be automatically detected when selecting bottom (dotted lines)

If you need to add more soft boundaries, select them after bottom selection.



To allow Start point definition outside the machining domain click: Start : Inside

In this case you must specify a clearance with respect to the pocket boundary.



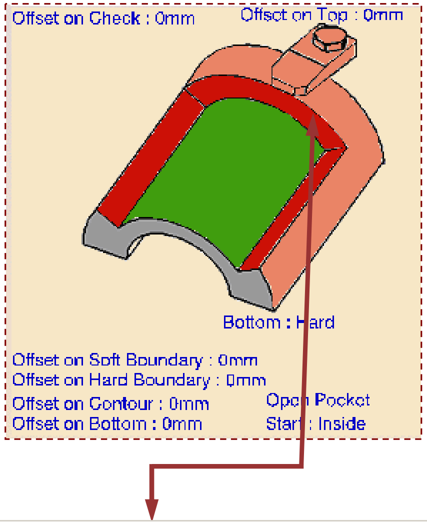
Start points are computed automatically for Closed Pockets.

Student Notes:

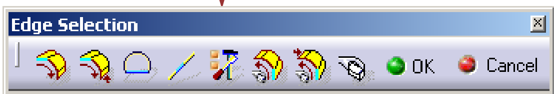
4-Axis Pocketing Operation: Geometry (3/3)

The pocket boundary must be closed. It can be defined by three ways:

- **If the Contour Detection contextual command is selected, click the pocket bottom. The boundary of the selected face will be proposed as pocket boundary.**



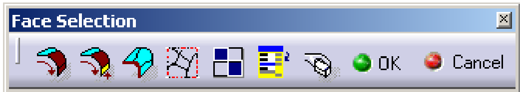
- **Otherwise, click guiding elements in sensitive dialog box. In this case the Edge Selection wizard appears to help you to define the edges of the pocket boundary or**



Edge Selection wizard

OR

- **Select 'By Boundary of Faces' in contextual menu. In this case the Face Selection toolbar appears to help you to define the faces of the pocket boundary.**



Face Selection wizard

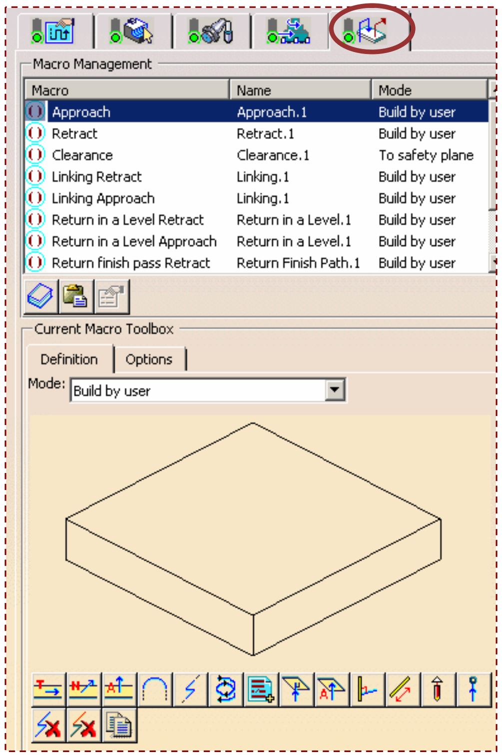
Student Notes:

Student Notes:

4-Axis Pocketing Operation: Macros

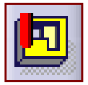
You can define transition paths in your machining operations by means of NC Macros. These transition paths are useful for providing approach, retract and linking motion in the tool path.

- **An Approach macro is similar to that of pocketing operation and used to approach the operation start point.**
- **A Retract macro is used to retract from the operation endpoint.**
- **A Linking macro may be used:**
  - **to link two non- consecutive paths**
  - **to access finish and spring passes.**
- **A Return on Same Level macro is used in a multi-path operation to link two consecutive paths in a given level.**
- **A Return between Levels macro is used in a multi-level machining operation to go to the next level.**
- **A Return to Finish Pass macro is used in a machining operation to go to the finish pass.**
- **A Clearance macro can be used in a machining operation to avoid a collision with the fixture.**



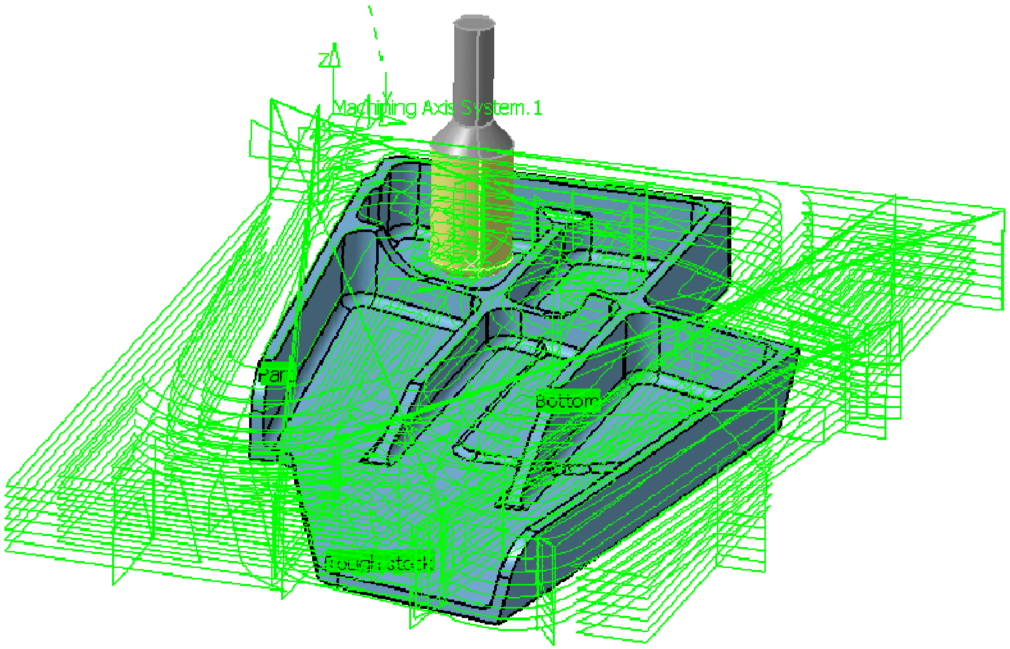
When a collision is detected between the tool and the part or a check element, a clearance macro is applied automatically. If applying a clearance macro would also result in a collision, then a linking macro is applied. In this case, the top plane defined in the operation is used in the linking macro.

Prismatic Roughing Operation



In this lesson, you will learn how to create a Prismatic Roughing Operation. A Prismatic Operation is used to quickly rough machine a part in a single operation.

- **Introduction**
- **Creating a Prismatic Roughing Operation**
- **Strategy**
- **Geometry**
- **Macros**



Student Notes:

Student Notes:

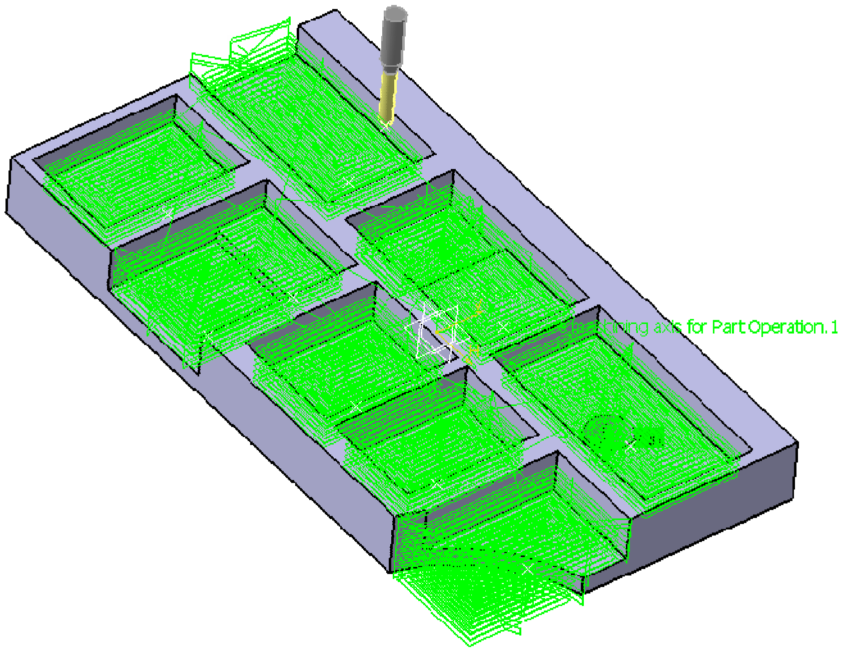
Prismatic Roughing Operation: Introduction

Prismatic Roughing Operation is particularly useful for parts that include drafted pockets or multiple bottom pockets, as CATIA automatically calculates stock in the inner pocket and provides input for next semi-finishing operation.

Prismatic Roughing Operation consists of:

- **Geometry considerations**
  - **Required Stock and Part**
  - **Optional check and limit element**
  - **Top, bottom and intermediate planes**
  - **Horizontal area detection**
  - **Offsets on Part, check and planes**

- **Machining strategy parameters**
  - **Helical strategy**
  - **Back & Force strategy**
  - **Machining mode by plane or by pocket**
  - **High Speed Milling capabilities**



Student Notes:

Creating a Prismatic Roughing Operation: General Process

1. **Type the Name of the Operation. (Optional because a default name is given by the system 'Type_Of_Operation.X')**
2. **Type the text of comment (optional)**
3. **Define operation parameters using the 5 tab pages**

Strategy tab page

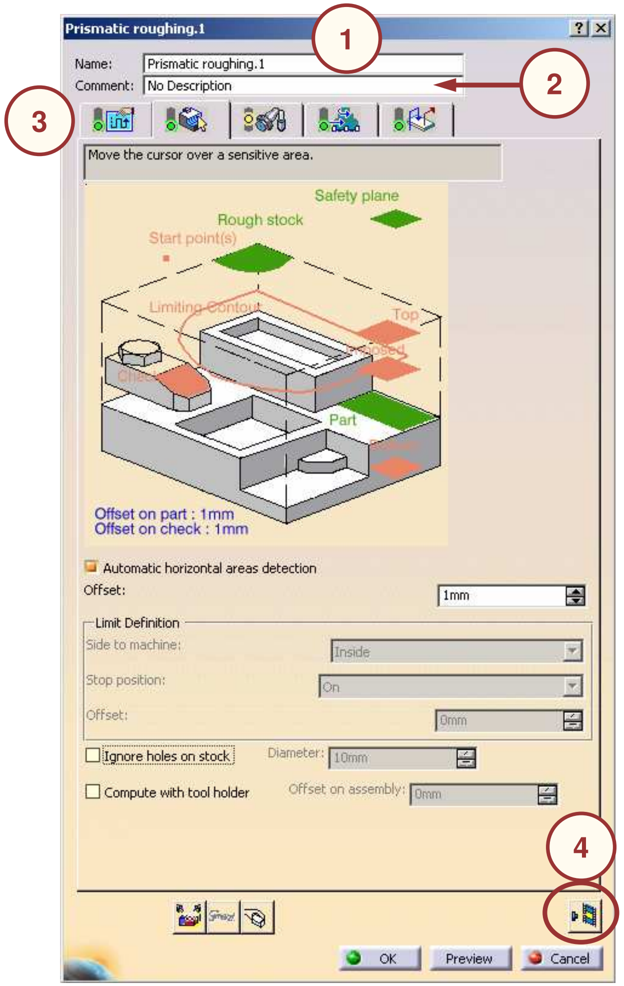
Geometry tab page

Tool tab page

Feeds & Speeds tab page

Macros tab page

4. **Replay and/or Simulate the operation tool path**

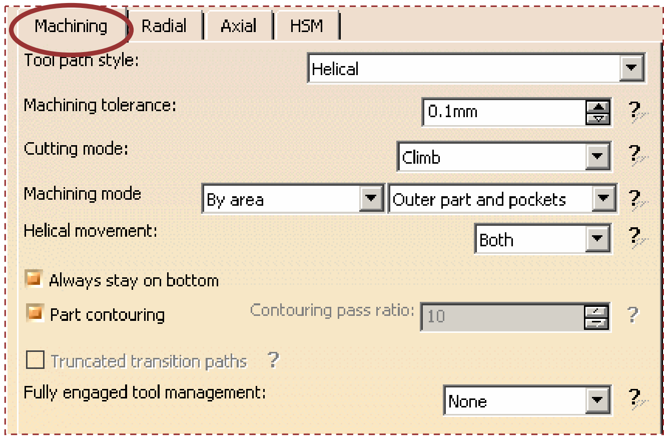


Student Notes:

Prismatic Roughing Operation: Strategy (1/5)

Machining Strategy Parameters:

Tool path style: Helical
The tool moves in successive concentric passes from the boundary of the area to machine towards the interior or from the interior to the boundary



Machining tolerance
Value of the maximum allowable distance between theoretical tool path and the tool path computed

Cutting mode
The cutting mode can be Climb or Conventional

Machining mode
The machining mode can be:
By plane: the whole part is machined plane by plane,

By area: the whole part is machined by area

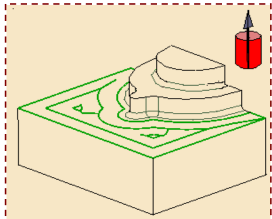
Outer part: only the outside of the part is machined

Pockets only: only pockets on the part are machined

Outer part and pockets: the whole part is machined external area by external area and pocket by pocket.

Helical movement
This option allow to define if Helical movement is Inward, Outward or Both (mixed between both strategies)

Helical



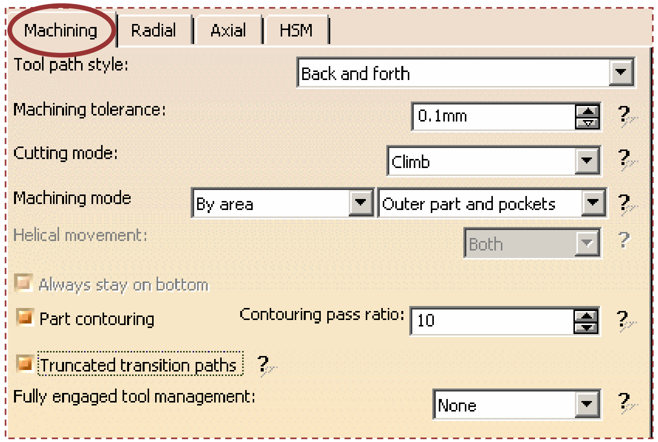
Always stay on bottom
When this option is checked, the linking path between two areas remains in the plane currently machined.

Student Notes:

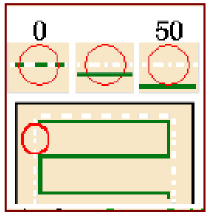
Prismatic Roughing Operation: Strategy (2/5)

Machining Strategy Parameters:

Tool path style Back and Forth
The tool moves following selected direction as Back and Forth

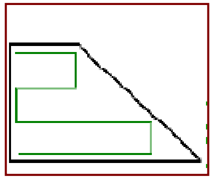


Part contouring
Allow to define a contouring pass with a dedicated ratio

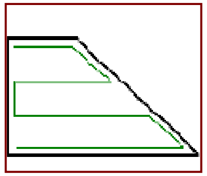


Truncated transition paths
Allow to optimize transition path on soft boundaries

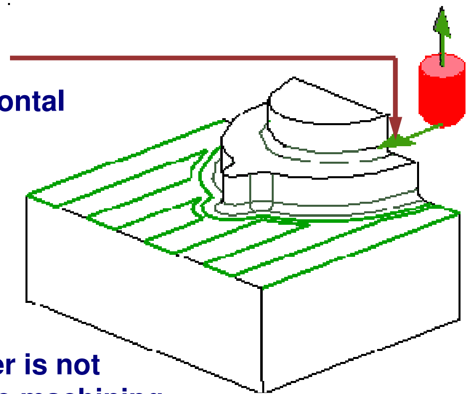
With activation



Without activation



Back and Forth
To select the machining direction, click the horizontal arrow



Fully engaged tool management:
Full material removal is detected in roughing of hard material, where the stepover is not always respected. This can be achieved by 'Trochoid' motions or by adding more machining planes using 'MultiPass'. Hence there is no danger of damage of tool.
The parameter is available for Tool path style 'Helical', 'Concentric' and Offset on Part.

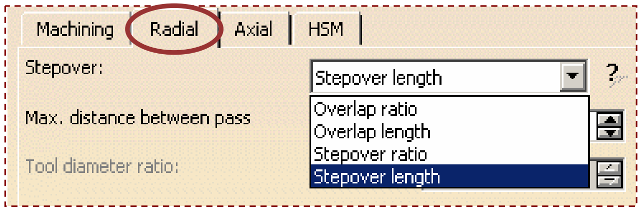
Student Notes:

Prismatic Roughing Operation: Strategy (3/5)

Radial Strategy Parameters:

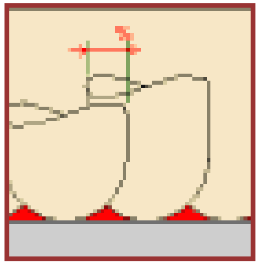
Four different ways to define distance between passes:

- Overlap ratio
- Overlap length
- Stepover ratio
- Stepover length

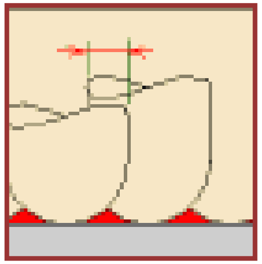


Overlapping
Define by a ratio (percentage of the tool diameter) or a length based on the side of the tool

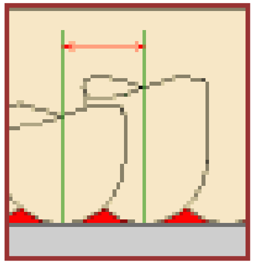
Stepover
Define by a ratio or a length based on tool axis



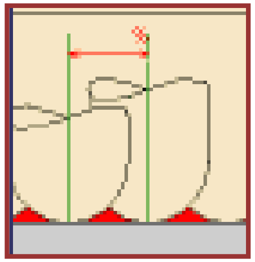
Overlap ratio



Overlap length



Stepover ratio



Stepover length

Student Notes:

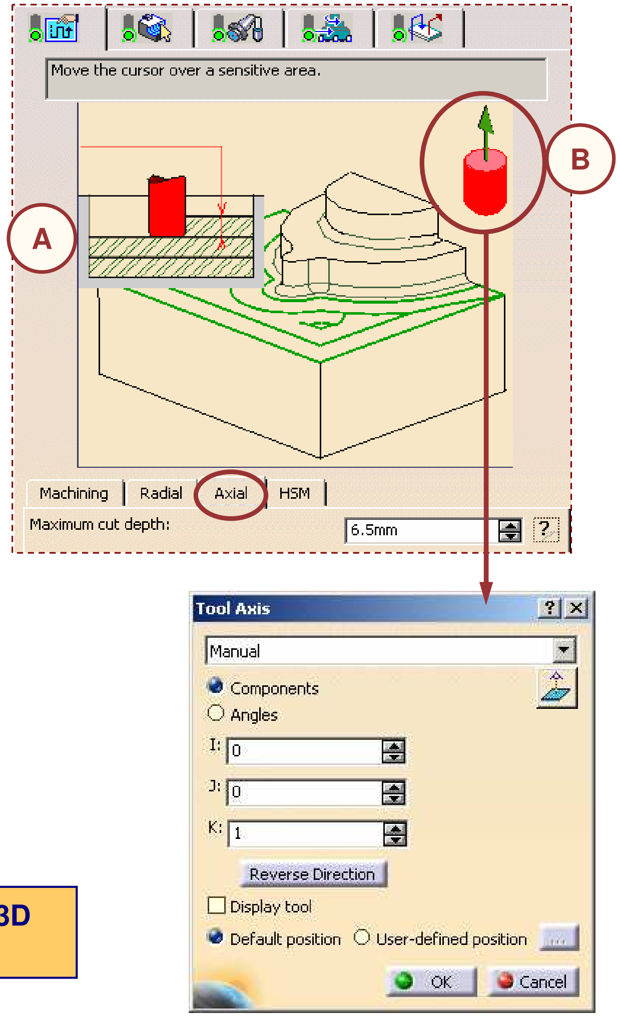
Prismatic Roughing Operation: Strategy (4/5)

Axial Strategy Parameters:

Maximum cut depth (A)
This parameter gives the maximum cut depth. The Cut depth is computed according to this maximum value and distance between top and bottom plane.

Tool axis definition (B)
Change the tool axis by selecting in the contextual menu which will display a dialog box where you can choose one of the following options:

- **Feature-defined**: select a 3D element such as a plane that will serve to automatically define the best tool axis.
- **Selection**: you select a 2D element such as a line or a straight edge that will serve to define the tool axis,
- **Manual**: type the XYZ coordinates,
- **Points in the view**: click two points anywhere in the view to define the tool axis,



You can reverse the tool axis direction and also obtain a real 3D representation of the tool in the 3D viewer.

Student Notes:

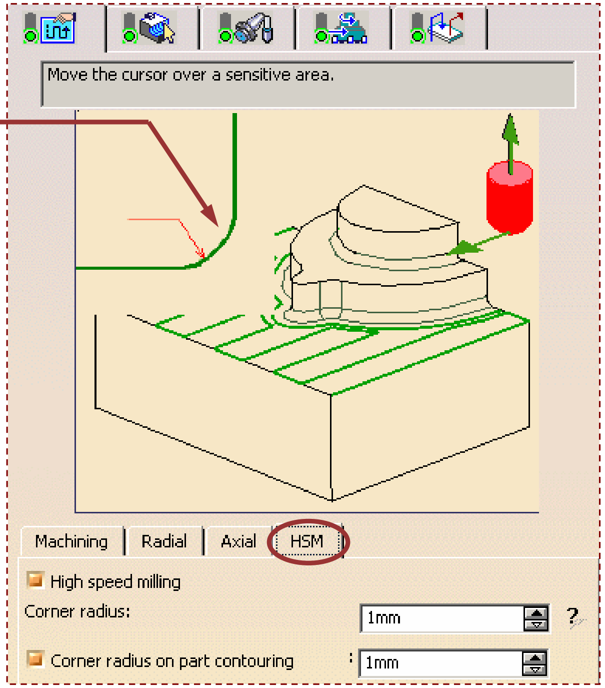
Prismatic Roughing Operation: Strategy (5/5)

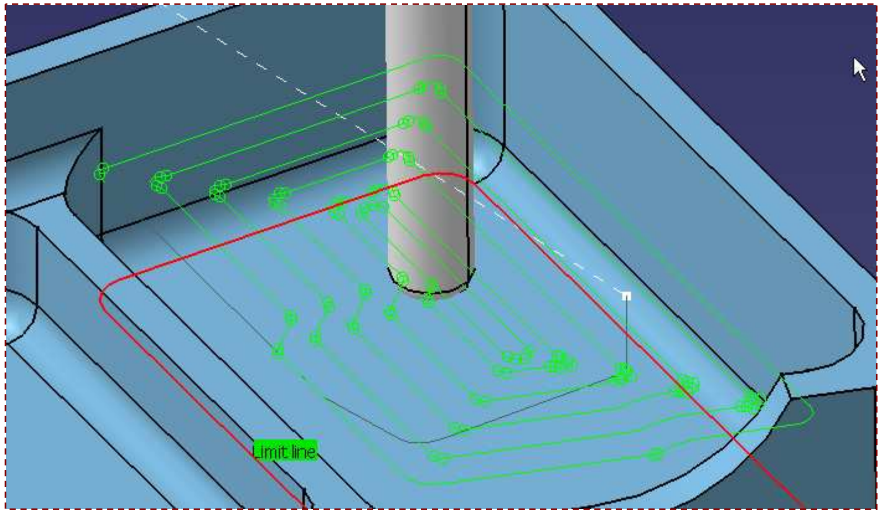
HSM Strategy Parameters:

High Speed Milling

- Corner radius parameter allows you to have a cornering tool path
- Corner radius on part contouring parameter allows you to specify a dedicated value of cornerization for the Part contouring path

Corner radius





Prismatic Roughing Operation: Geometry (1/3)

Student Notes:

Geometry Parameters:

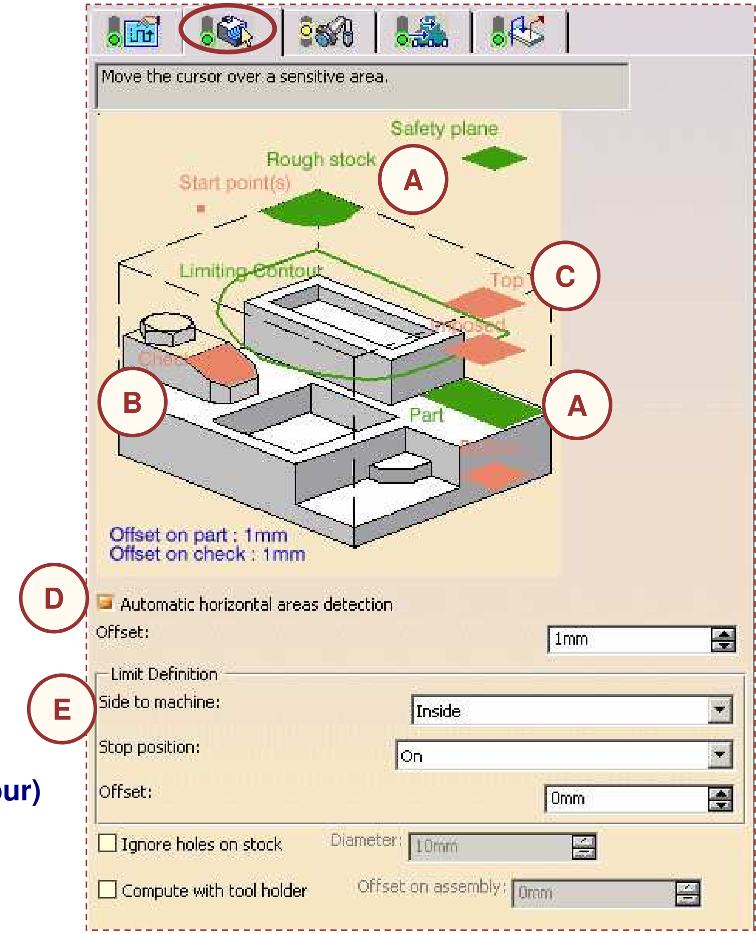
A. **Rough stock and Part**
   Roughing operation will remove all material between the Rough stock and final part

B. **Check**
   Element to avoid during machining

C. **Top and Bottom planes**
   If you want to limit the height of the machining select top and/or bottom plane

D. **Automatic horizontal areas detection**
   Use this option to automatically detect horizontal areas on the part and to apply a different offset on these areas

E. **Limiting contour**
   - **Use this option to restrict the machining area.**
   - **Define closed contour with Edge Selection wizard**
   - **Specify side to machine (inside or outside) and Stop position (how the tool has to stop according this contour)**



Student Notes:

Prismatic Roughing Operation: Geometry (2/3)

Geometry Parameters:

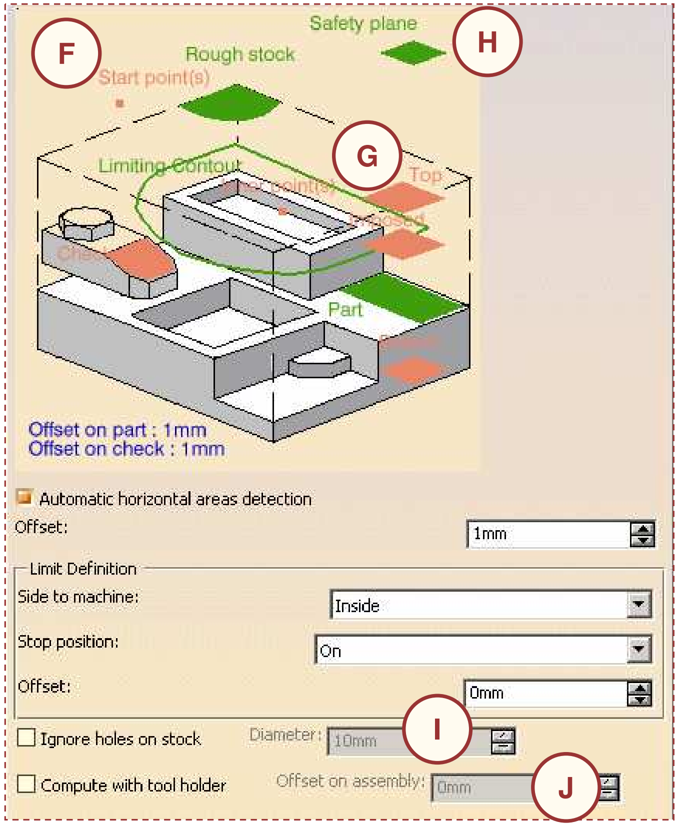
F. Start point(s)
Only for open area (no pocket). Defined point must not be in collision with Part or Stock.

G. Inner point
Only for closed area (pocket) using Drilling approach macro.

H. Safety plane
By default, all linking paths are using safety plane for retract and approach motion. If optimise retract option is ON, safety plane will be used only for first approach and last retract motions.

I. Ignore holes on stock
When you select the check box Ignore holes on stock, holes on the rough stock are ignored. Then you can define the diameter under which holes are to be Ignored.

J. Compute with tool holder
You can compute the tool path by selecting this option to avoid collisions with the tool holder.
When this check box is selected, you can define an offset on the tool holder assembly.
When this check box is cleared, the tool path is computed only with the tool.

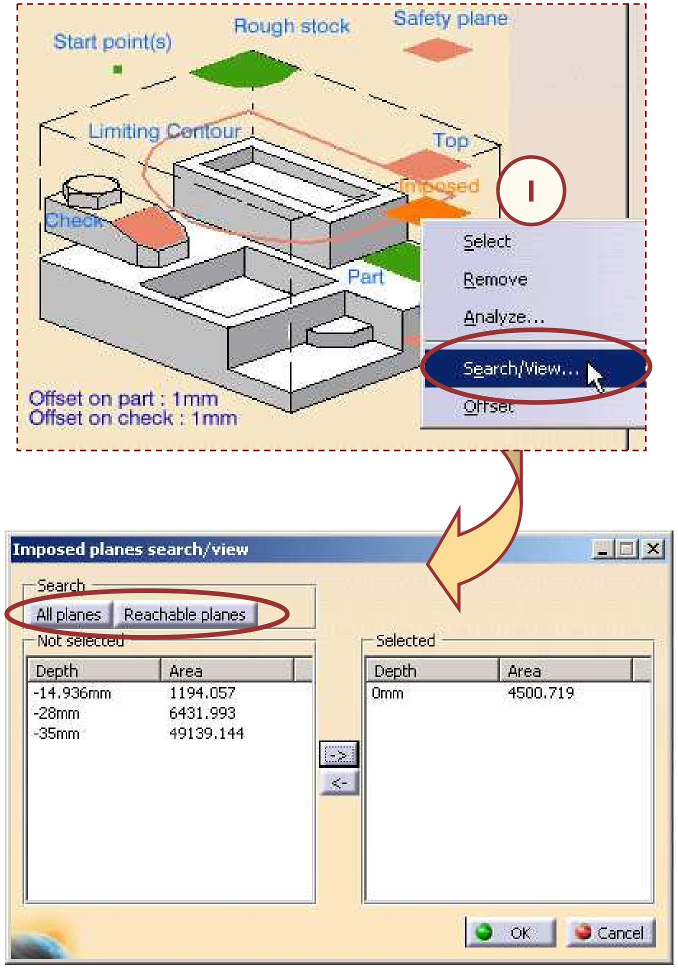


Student Notes:

Prismatic Roughing Operation: Geometry (3/3)

Geometry Parameters:

I. **Imposed planes**
Allows you to find all planar surfaces in a part then select among them imposed surfaces.



Search is done on:
all of the planar surfaces in the part, or only the planes that can be reached by the tool you are using (small pocket, counter-draft area are skipped)

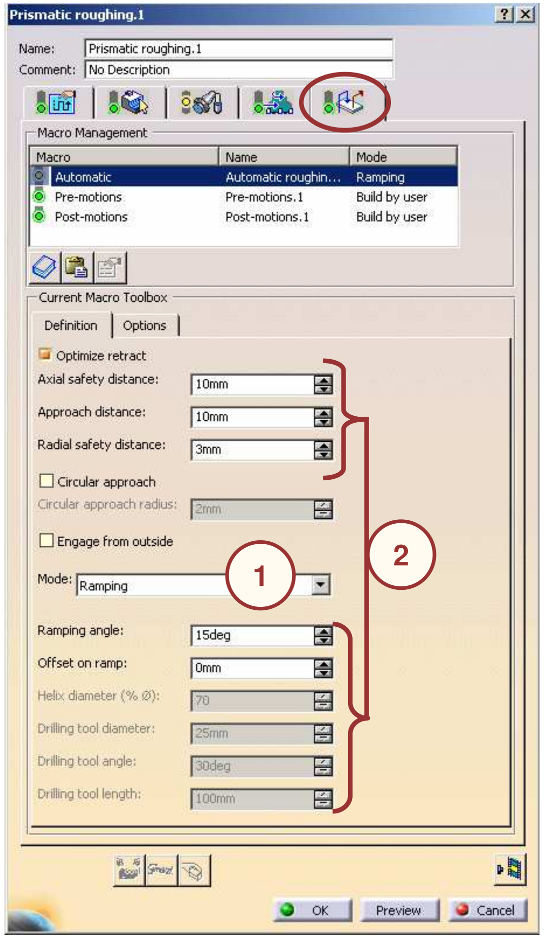
You can set an offset on those planes.

Offset on imposed planes has to be greater than the global offset on part. Otherwise it will not be respected.
Use Automatic horizontal areas detection to manage an offset smaller than the global one.

Student Notes:

Prismatic Roughing Operation: Macros

1. Under Mode item, you can select among:
   - Plunge: the tool plunges vertically,
   - Drilling: the tool plunges into previously drilled holes. You can change the drilling tool diameter, angle and length
   - Ramping: the tool moves progressively down at the ramping angle,
   - Helix: the tool moves progressively down at the ramping angle with its center along a (vertical) circular helix of Helix diameter.
   - Radial Only: When drilling holes exist, you can define start points and use Radial Only to avoid any plunge or ramping macro.
2. According to your Approach mode, you can modify the default parameters



Using Optimize retract button, you optimize tool retract movements. This means that when the tool moves over a surface where there are no obstructions, it may not rise as high as the safety plane because there is no danger of tool-part collisions. The result is a gain in time.

The Radial safety distance is the maximum distance that the tool will rise to when moving from the end of one pass to the beginning of the next one.

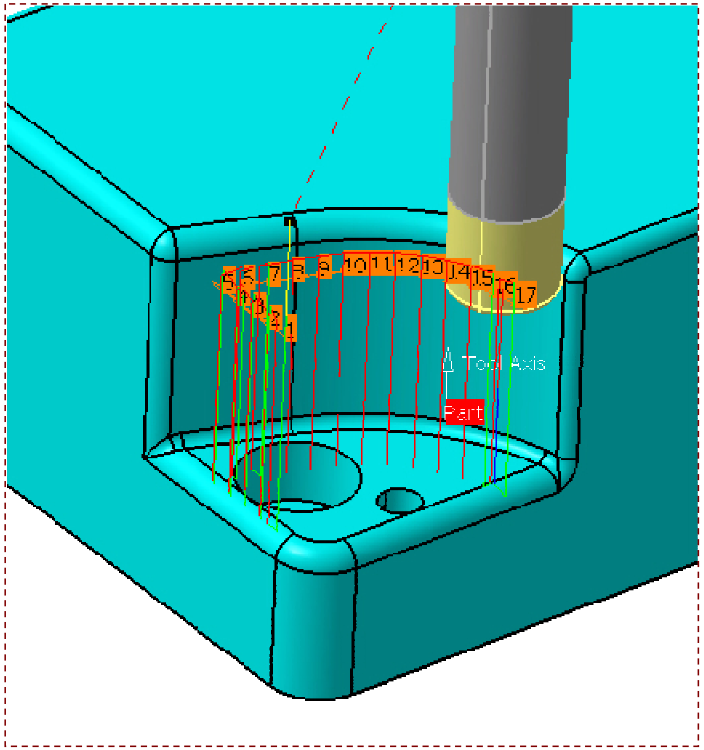
Using Circular approach, you can insert a cornering arc in the approach movements and define the radius for the approach.

For open pockets select the Engage from outside check box to create engagements from external zones.

Student Notes:

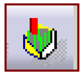
Plunge Milling

In this lesson, you will learn how to create a Plunge Milling Operation.



Student Notes:

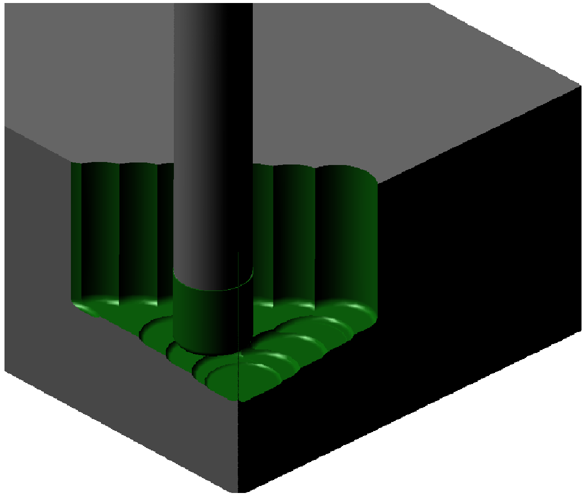
Plunge Milling: Introduction



Plunge milling operation is used to rough machine a part by plunging the tool into the material. The operation is useful to easily machine deep cavities and the machining time is drastically reduced.

The plunge cutter approaches the material from above, penetrates to maximum depth, and withdraws to step over for the next plunge. The main cutting forces on the tool and machine spindle are axial. No side forces to bend the tool. Hence the machine can maintain high speeds and feeds straight to the bottom of deep cavities.

Special Plunge tools or Plungers are used to perform the Plunge Milling Operation.



Plunge Milling offers:

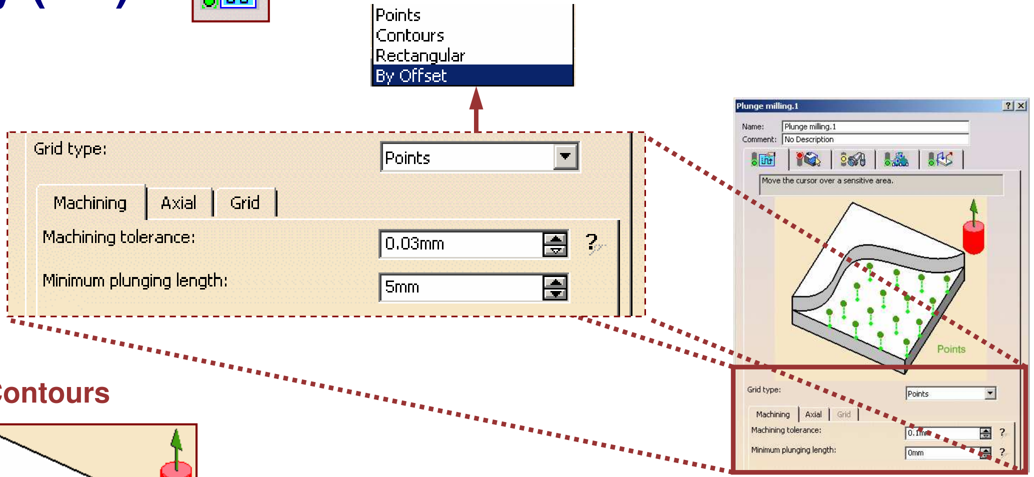
- High material removal rate.
- Principally Z-axial cutting force reduces power consumption due to lower cutting forces.
- System extremely efficient in all roughing operations and deep cavities.
- Process minimizes tool deflection and side forces.

Plunge milling can be the only possible solution for long tool overhangs and unstable conditions. It is a highly productive method for internal milling of deep cavities and milling externally along deep shoulders.

Care must be taken that there is a clear path for chip takeaway as the chips will build up fast.

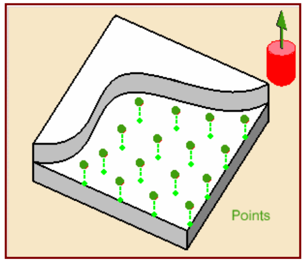
Student Notes:

Plunge Milling: Strategy (1/5)



The four Grid types are:

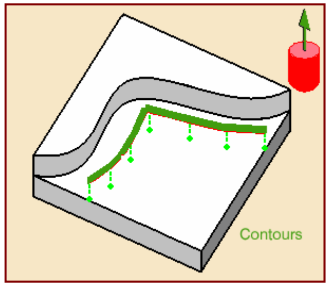
Points



It enables to select points in the 3D viewer with the selection trap.

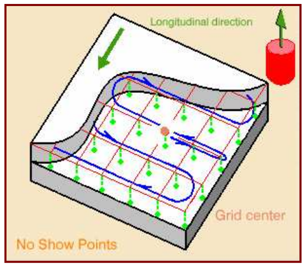
'Renumbering' and 'Automatic ordering' of selected points is possible by contextual menu.

Contours



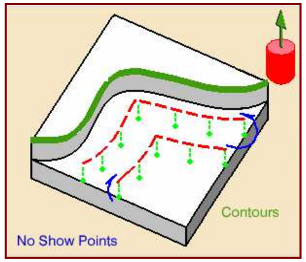
It gives access to the edge selection wizard. Each contour is taken into account and can be reoriented.

Rectangular



Grid Center selection gives access to the selection of a point which will be taken into account to define the grid position.

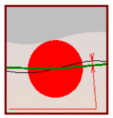
By Offset



By Offset gives computation of several contours by the offset value from the selected contour.

Machining Tolerance

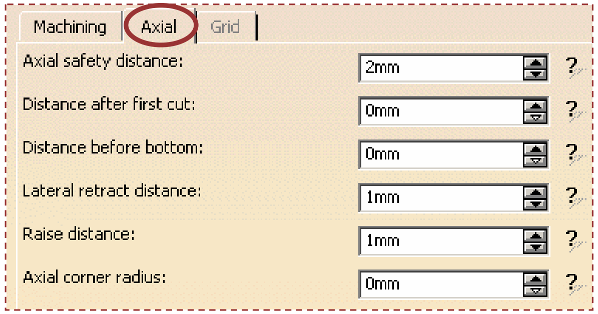
Value of the maximum allowable distance between theoretical tool path and the computed tool path

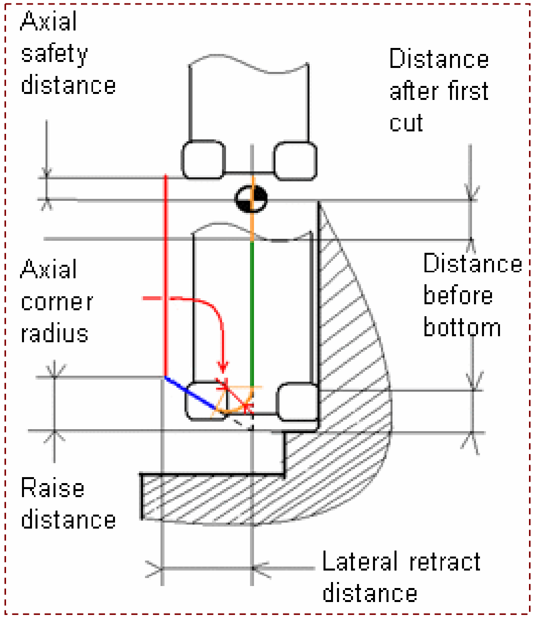


Plunge Milling: Strategy (2/5)

Axial parameters:

- **Axial safety distance:**
  It is the distance from which the tool starts approaching the stock. The start point of the plunging is derived from the value of this distance.
- **Distance after first cut:**
  It is the distance at which the tool starts plunging in the stock. The plunge feedrate is taken into account for this distance.
- **Distance before bottom:**
  It is the distance at which the tool moves with plunge feedrate to reach the bottom. Thus it gives facility to use lower feedrate at the end of plunging for finishing.
- **Lateral retract distance:**
  It is the lateral distance by which the tool retracts from the material after completion of plunge operation. The tool moves in both X & Y directions as per the values.
- **Raise distance:**
  It is the distance by which the tool retracts in Z direction. The tool moves with retract feedrate.
- **Axial corner radius:**
  It is the radius which gives better control on the movement of the retract motion. A larger Axial corner radius will remove more material on the bottom.





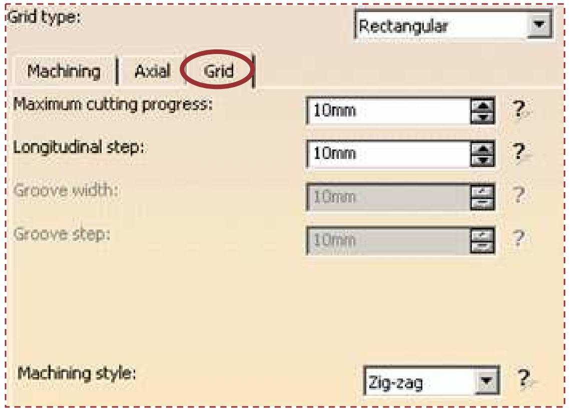
Student Notes:

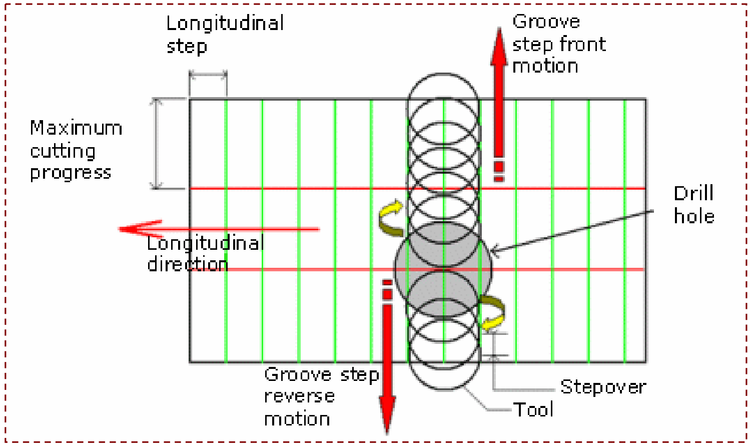
Student Notes:

Plunge Milling: Strategy (3/5)

Grid parameters:

- **Maximum cutting progress:**
  It is a maximum cutting progress distance in transverse direction.
- **Longitudinal step:**
  It is a tool stepover distance in longitudinal direction.
- Grid is formed with definition of maximum cutting progress and longitudinal step values. The tool moves to the points of the grid for plunging.
- Starting from the Grid center, the grid is computed along a Longitudinal direction. The tool moves first in order to machine a groove and then reach each of the points of the grid with a constant fixed order defined by a machining style.
- Default groove width value is the tool diameter. If groove width is greater than twice the tool diameter, then Zig-zag machining style must be used.





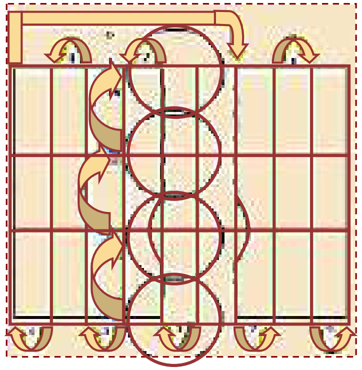
Student Notes:

Plunge Milling: Strategy (4/5)

Grid parameters:

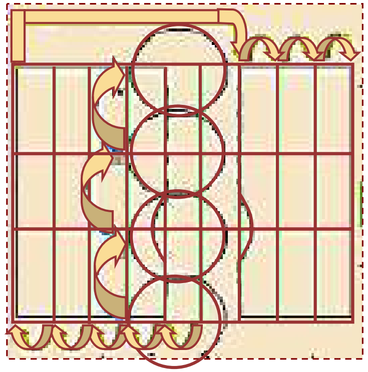
The two Machining styles are:

Zig-zag

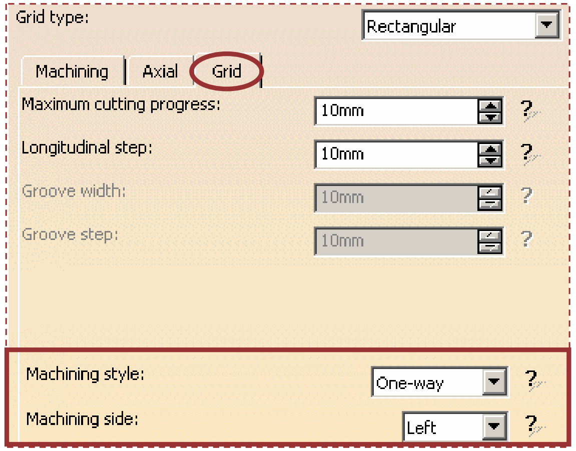


The tool path alternates directions during successive passes.

One-way

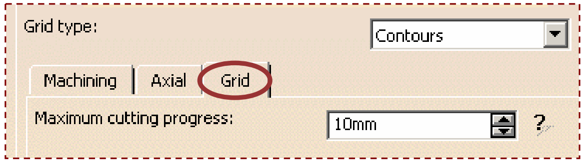


The tool path always has the same direction during successive passes and goes diagonally from the end of one tool path to the beginning of the next.



‘Left’ or ‘Right’ Machining side gives side for the start of next tool path step. Material is on the selected side.

A set of contours is defined and may be ordered or not, closed or not. Using Maximum cutting progress i.e. discretization step, the contact points are computed from the contours.

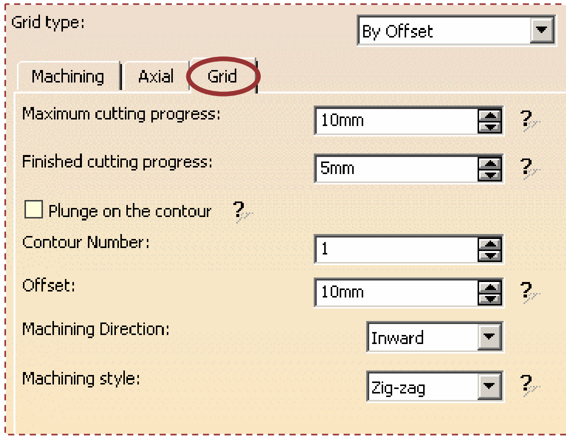


Student Notes:

Plunge Milling: Strategy (5/5)

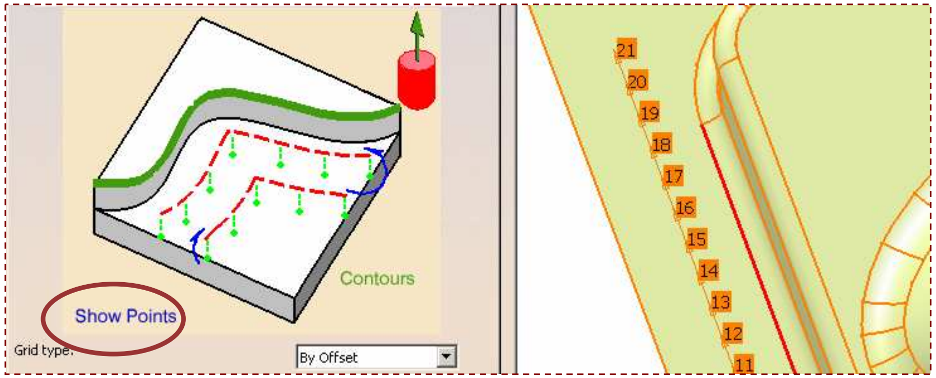
Grid parameters:

- **Finished cutting progress:**
  It is the step on the initial groove along the direction of the contour.
- **Plunge on the contour:**
  It allows to plunge on the contour.
- **Contour Number:**
  It specifies the number of contours.
  It should be at least one.
- **Machining Direction:**
  It specifies the direction for milling i.e. inward or outward.



- **Offset**
  It specifies the value of offset between number of contours specified.

You can visualize the plunging points before the computation of the operation.

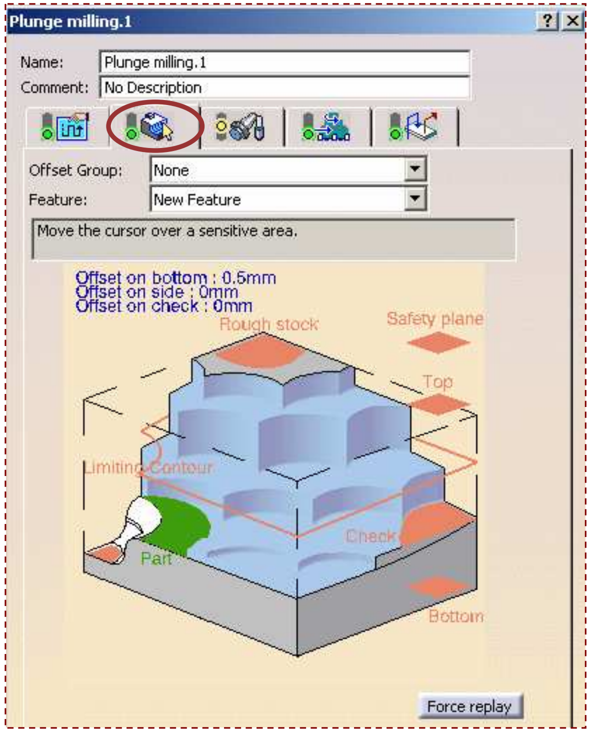


Student Notes:

Plunge Milling: Geometry

Geometry parameters:

- **Part to machine (mandatory)**
- **Offset on bottom (horizontal area), side (vertical area) or check element (usually clamp).**
- **Rough stock (optional): If the rough stock is not defined, the rough stock defined at the PO level is taken into account to compute the remaining material after all the operations performed before this operation. Thus operation uses the rework technology.**
- **Top: All machining above this plane will not be taken into account.**
- **Bottom: The plunging movement will stop at this level, if reached before the part.**
- **Safety plane: The tool rises to this plane at the end of the tool path. Thus tool collision with the part is avoided.**
- **Limiting Contour: It is used in case of rectangular grid selection in order to keep the points situated inside the limiting boundary.**



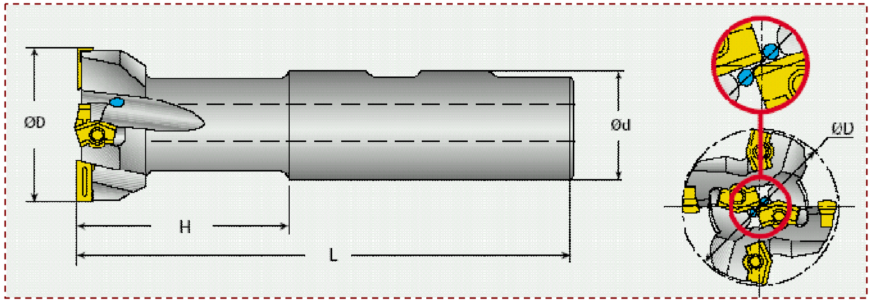
Don't forget to select 'Force Replay' button to update the 'actual stock' if needed.

Student Notes:

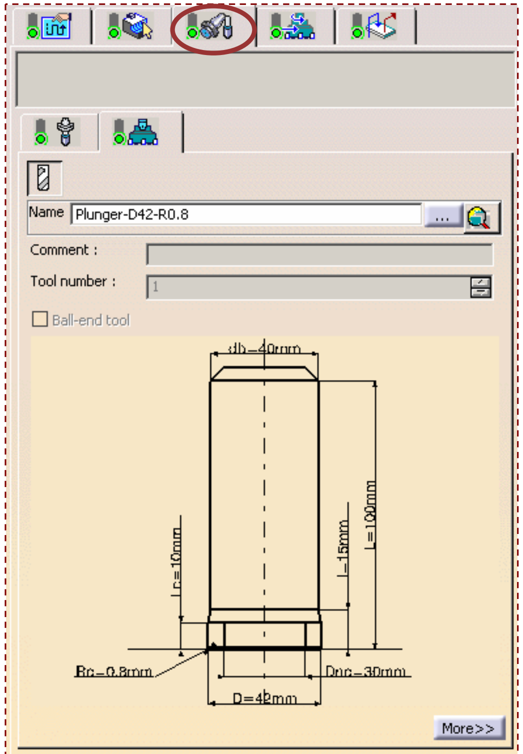
Plunge Milling: Tools

Plunge milling cutters are designed for high metal removal rates.

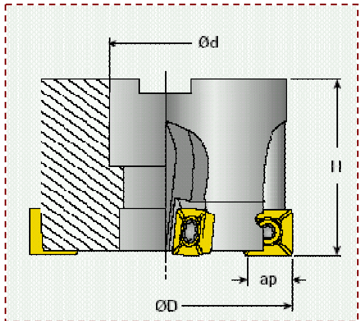
Center cutting plungers:



Plunge milling in pockets without drilled hole is possible only with Centre cutting plungers. The inserts mounted at the tip of the tool allows the tool to move same as a drill.



Side plunging milling cutters:



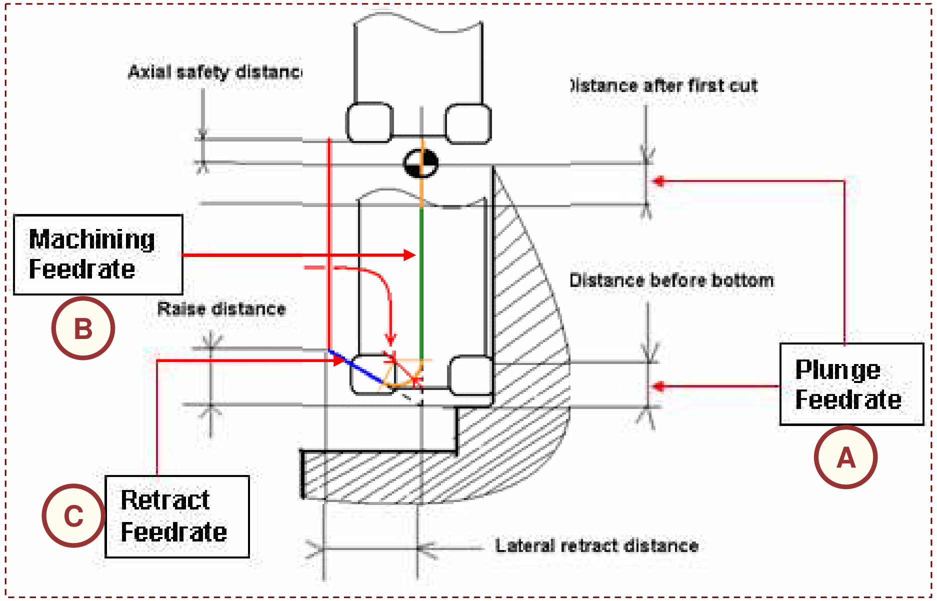
Side plunging milling cutters are used for plunge milling in external areas. Inserts mounted on periphery withstands high axial loads. Chip evacuation is effectively done.

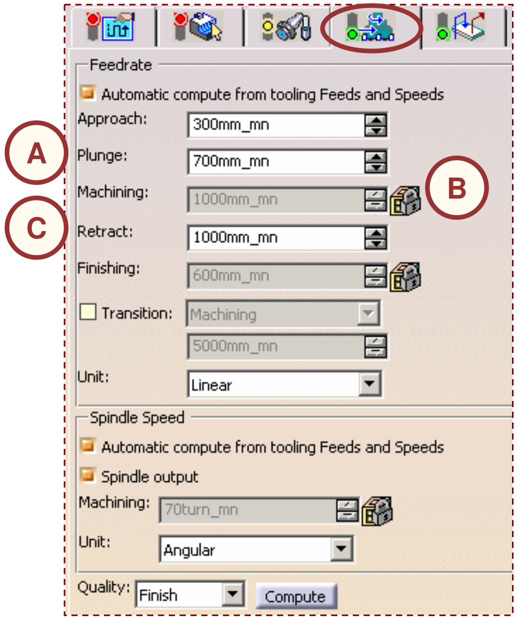
Plunge Milling: Feeds and Speeds

In 'Feeds and Speeds' tab, feedrates for approach, plunge retract, machining and finishing can be specified. Machining spindle speed also can be defined.

These Feedrate and Spindle Speed values can be defined either in Linear or Angular units.

The feedrates values of Plunge, Machining, Retract and Finishing are taken into account in the definition of Axial parameters.





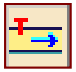
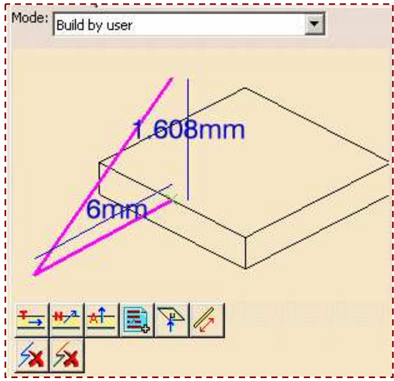
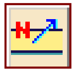
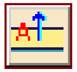
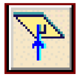
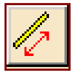
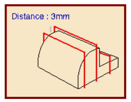
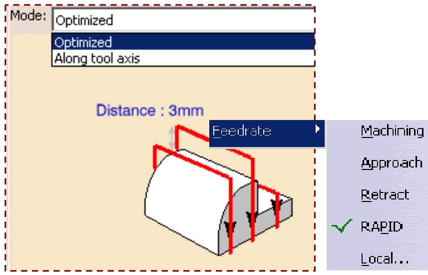
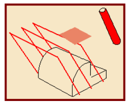
Feeds and speeds of the operation can be updated automatically according to tooling data and the Rough or Finish quality of the operation.

Student Notes:

Student Notes:

Plunge Milling: Macro

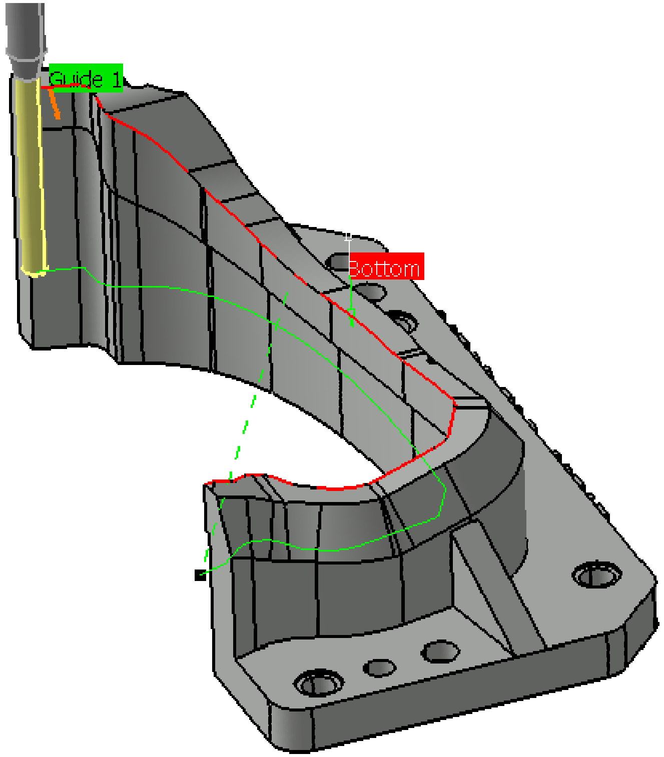
For Start and End Points, Approach and Retract Macros need to be defined.
Clearance macro can be used to define the horizontal path between two machining positions.
You can modify the feedrate of the clearance macro through contextual menu.

Type Of Motion	Mode	Type Of Macro	Icon	Image Elaborating Macro
Approach	Build by user	Add Tangent Motion		
		Add Horizontal motion		
Retract		Add Axial motion		
		Add PP word list		
		Add motion perpendicular to a plane		
		Add distance along a line motion		
Clearance	Optimized	Optimized		
	Along tool axis	Along tool axis		

Profile Contouring Operation

In this lesson, you will learn how to create a Profile Contouring Operation.

- Introduction
- Creating a Profile Contouring Operation
- Strategy
- Geometry
- Feeds and Speeds

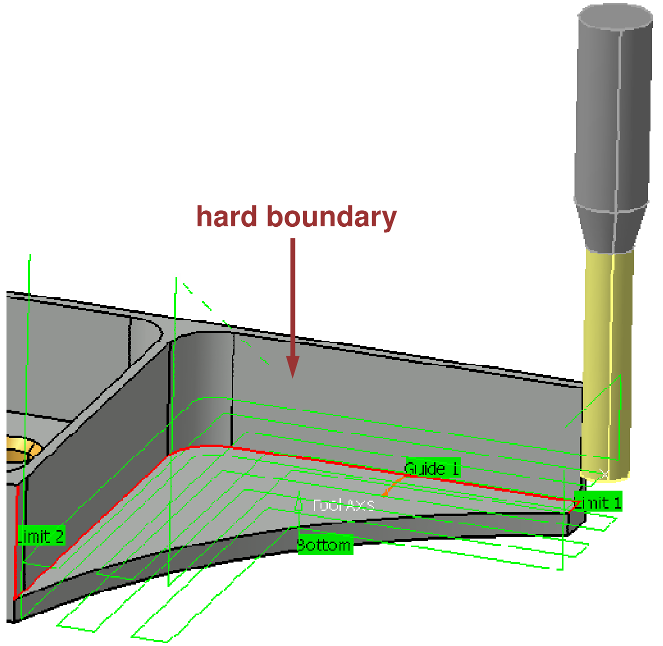


Student Notes:

Student Notes:

Profile Contouring Operation: Introduction

- A Profile Contouring Operation consists in cutting material along a hard boundary.
- The hard boundary may be either open or closed.
- Along axial direction, the material will be removed from the top to the bottom in one or several cuts.
- Along radial direction, the material will be removed by approaching the hard boundary in one or several parallel paths.
- The area is machined in One-way or in Zig-zag style.
- The Profile Contouring Operation can be performed:
    - between two planes
    - between two curves or
    - between a curve and surfaces



Student Notes:

Creating a Profile Contouring Operation: General Process

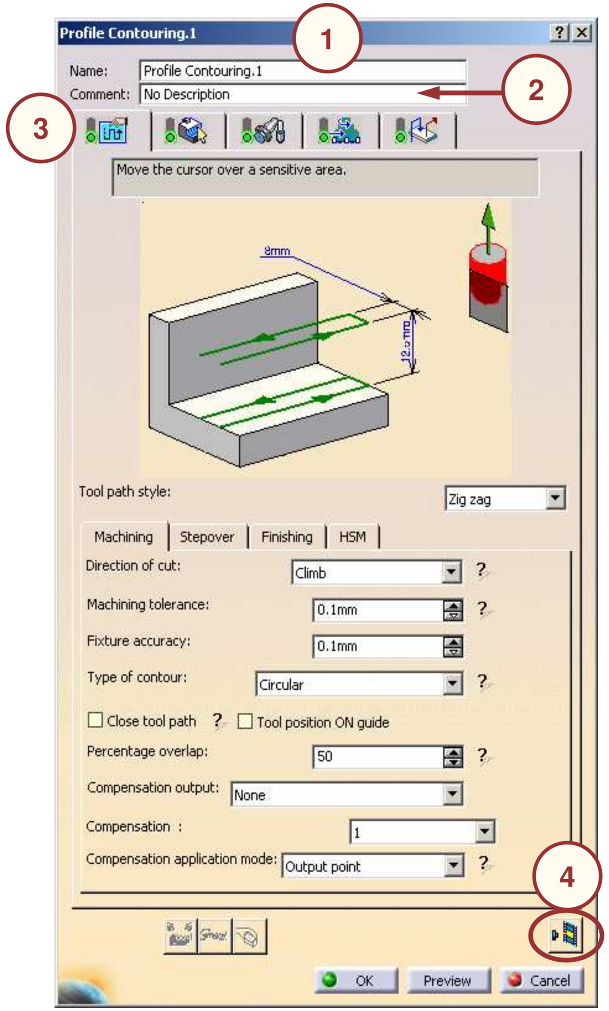
1. Type the Name of the Operation. (optional because a default name is given by the system 'Type_Of_Operation.X')

2. Type the text of comment (optional)

3. Define operation parameters using the 5 tab pages

- Strategy tab page
- Geometry tab page
- Tool tab page
- Feeds & Speeds tab page
- Macros tab page

4. Replay and / or Simulate the operation tool path



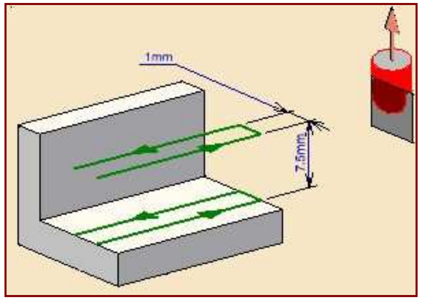
Student Notes:

Profile Contouring Operation: Strategy (1/6)

The Tool path styles for a Profile Contouring Operation are:

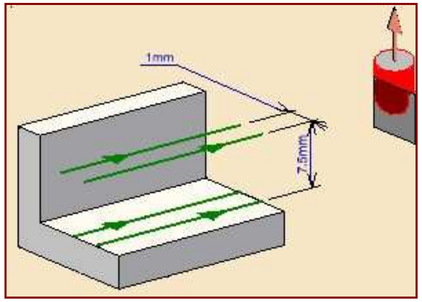
Zig - zag:

The tool alternatively machines in one direction and then in opposite direction.



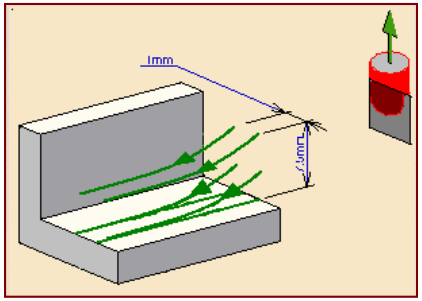
One way:

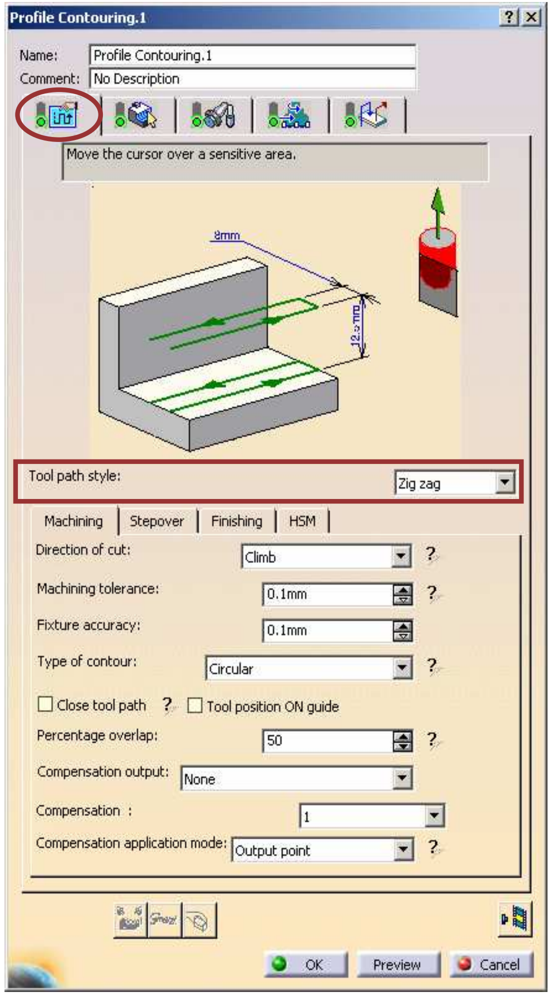
The tool machines always in the same direction.



Helix:

The tool machines maintaining constant tool contact with the part.





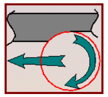
Student Notes:

Profile Contouring Operation: Strategy (2/6)

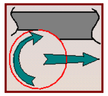
Machining Parameters:

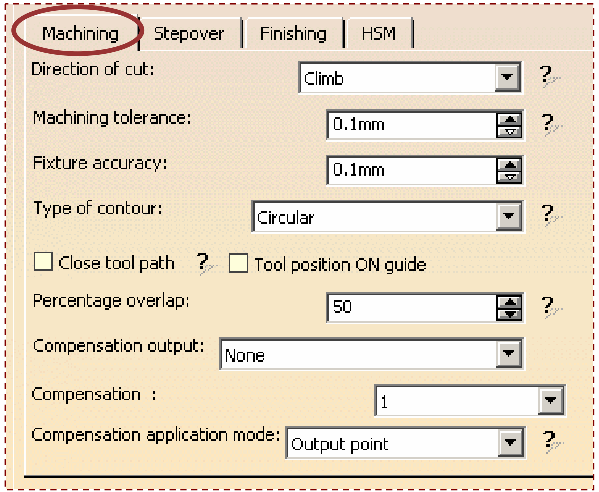
Direction of Cut

Climb: The front of the advancing tool cuts into the material first



Conventional: The back of the advancing tool cuts into material first





Machining Tolerance

Value of the maximum allowable distance between theoretical tool path and computed tool path

Fixture Accuracy

Local machining tolerance for fixtures

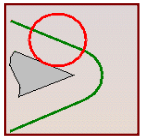
Type of Contour:

Circular: The tool pivots around the corner point, following a contour whose radius is equal to the tool radius

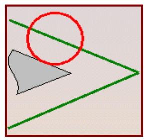
Angular: The tool does not remain in contact with the corner point, following a contour consists of line segments

Optimized: The tool follows a contour derived from the corner that is continuous in tangent

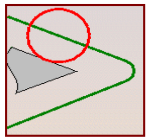
Forced Circular: The tool follows a near-circular contour consistd of line segment



Circular



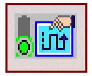
Angular



Optimized

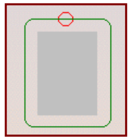
Student Notes:

Profile Contouring Operation: Strategy (3/6)



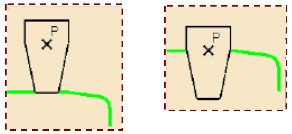
Machining Parameters:

Close tool path:
Option to machine the complete contour of a closed area.



For the Helix tool path style, 'Close tool path' and 'Percentage overlap' options are deactivated.

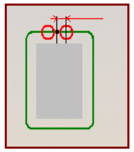
Tool position ON guide:
Specifies the position of the tool tip on the guiding elements.



Compensation:
Number of the tool compensation. It must be a number available on the tool used for the operation.

Compensation application mode:
You have to choose if compensation is applied on the tool output or guiding point.

Percentage overlap:
When « close tool path » is active, this is the overlap at the end of the tool path, expressed as a percentage of the tool diameter.

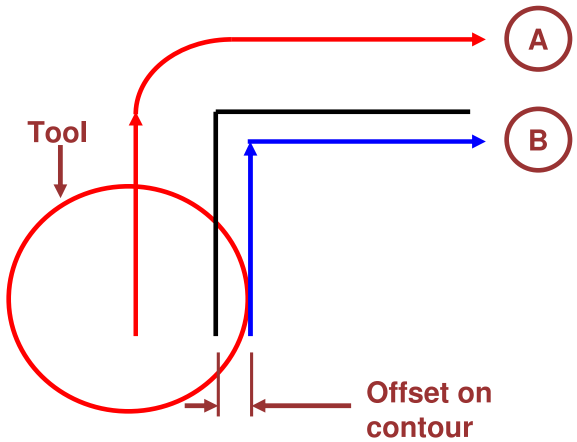


Compensation output:
Allows you to manage the generation of cutter compensation (CUTCOM) instructions in the NC data output in Between Two Planes machining mode.

A **2D radial tip:**
In the generated code, the toolpath is defined by the tool tip trajectory

B **2D radial profile:**
In the generated code, the toolpath is defined by the contact point trajectory

Compensation output

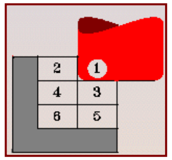


Student Notes:

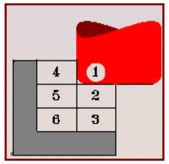
Profile Contouring Operation: Strategy (4/6)

Stepover Parameters:

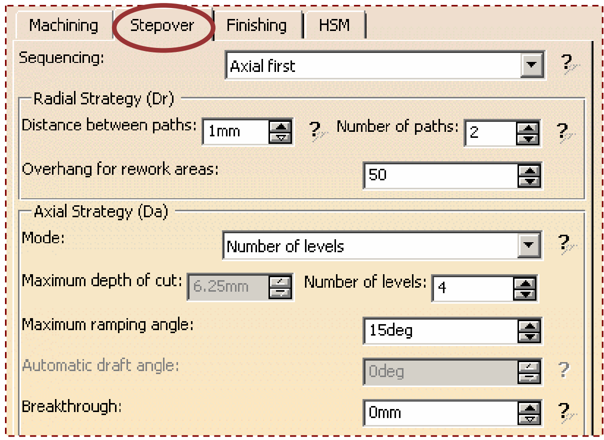
Sequencing:



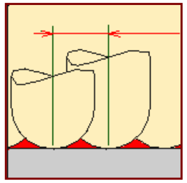
Radial first



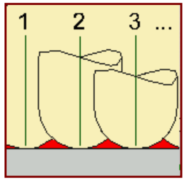
Axial first



Radial Strategy:



Distance between paths:
It is the distance between two radial paths.



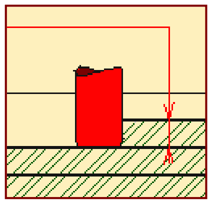
Number of paths:
It is the total number of radial paths.

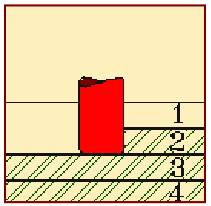
Axial Strategy:

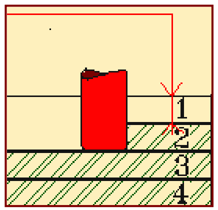
Max depth of cut
The maximum distance between two levels

Number of levels
The number of levels from the top to the bottom

Number of levels without top
The bottom, the number of levels and the depth of cut.

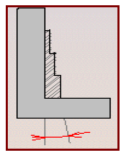




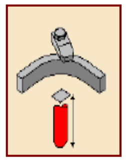


Maximum ramping angle (for Helix)
You can specify multiple radial passes with control of maximum ramping angle and depth of cut.

Automatic draft angle
Incremental increase of thickness on flank (not available with Helix)



Breakthrough
Only in soft bottom.
It is an offset in order to specify a virtual bottom.

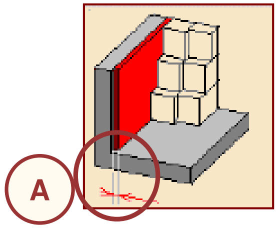


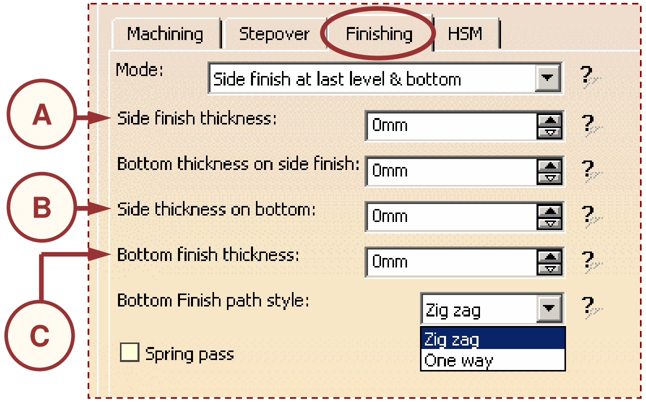
Student Notes:

Profile Contouring Operation: Strategy (5/6)

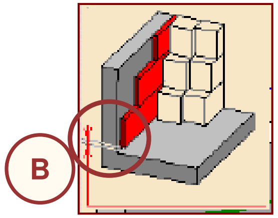
Side Finish Pass mode:

At last level:
activate a radial finish pass only at last level.



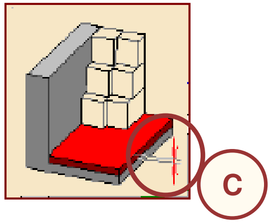


At each level:
Activate a radial finish pass at each level
(not available for Helix tool path style.)



Bottom Finish Pass mode:

At bottom:
Specify the thickness used for the bottom finish pass



Bottom Finish Path style:

Defines the bottom finish path style:
Available only for Zig zag or One way
This option is deactivated for Helix.

Spring Pass:
Duplicates last finish pass to compensate the spring of the tool.

The Finishing Feedrate will be used to cut the material on the Side and Bottom finish passes

Student Notes:

Profile Contouring Operation: Strategy (6/6)

HSM Parameters:

HSM is a capability to round corners in the tool path.

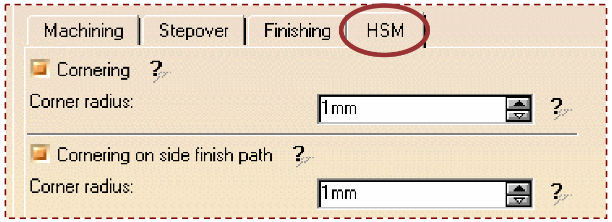
Cornering for HSM is available for Roughing and Finishing passes in the following guiding modes: Between two planes, Between curve and surfaces and Between two curves.

Cornering applies to inside corners for machining or finishing passes. It does not apply to:

- **Outside corners (for example, produced by angular or optimized contouring mode).**
- **Macros or default linking and return motions.**

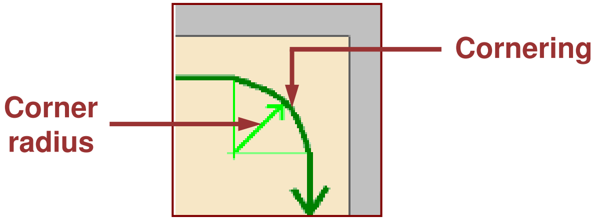
Cornering:
Specifies whether or not cornering for HSM is to be done on the trajectory.

Corner radius:
Specifies the radius used for rounding the corners along the trajectory of a HSM operation. Value must be smaller than the tool radius.



Cornering on side finish path:
Specifies whether or not tool path cornering is to be done on the side finish path.

Corner radius on side finish path:
Specifies the corner radius used for rounding the corners along the side finish path of a HSM operation. Value must be smaller than the tool radius.



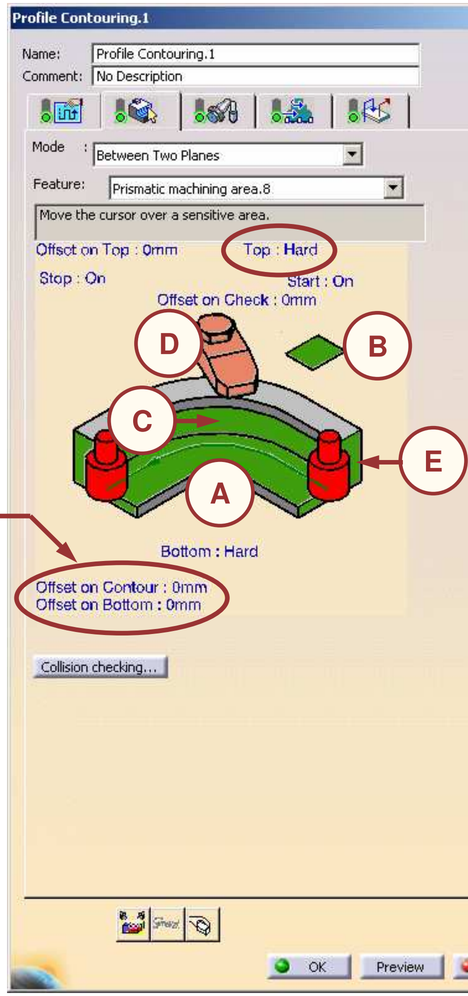
Student Notes:

Profile Contouring Operation: Geometry (1/4)

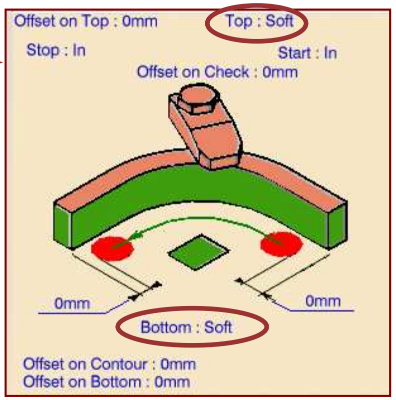
This tab page includes a sensitive icon dialog box that allows the selection of:

- **A** — **Bottom Plane**
- **B** — **Top Plane (for Multi-Levels operations only)**
- **C** — **Guiding Elements**
  Discontinuous contour is possible, allowing to machine several contours in one single operation thus providing better support for thin wall finishing.
- **D** — **Check Elements (Optional)**
- **E** — **Limiting Element (Optional)**

Offsets can be applied on the Top Plane, Bottom Plane, Contour, Check and Relimiting Elements.



Bottom and Top Soft:
User can switch bottom and top planes in Hard/Soft mode.



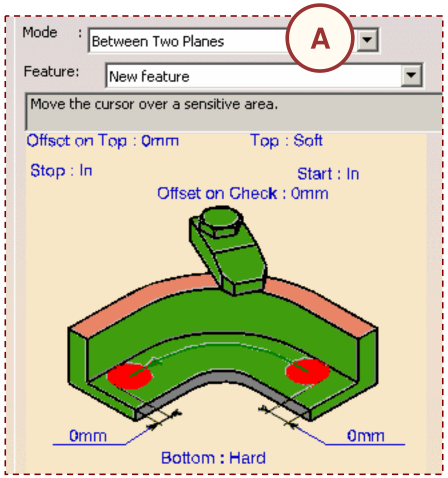
Only one closed guiding element (edges or sketch) must be used for Helix tool path style.

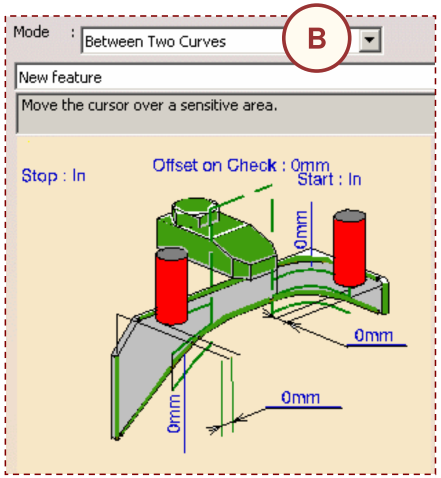
Student Notes:

Profile Contouring Operation: Geometry (2/4)

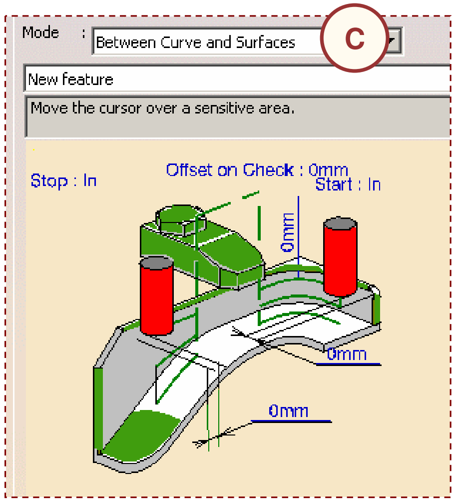
Profile Contouring Mode:

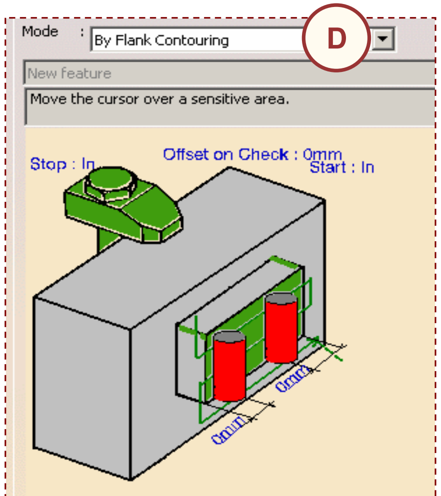
- **A** Between Two Planes
- **B** Between Two Curves (first curve needed, second is optional)
- **C** Between Curve and Surfaces
- **D** By Flank Contouring





Automatic point detection
In mode A, B and C this option will automatically detect extremities of guiding element (vertex) and assign them as re-limiting element.





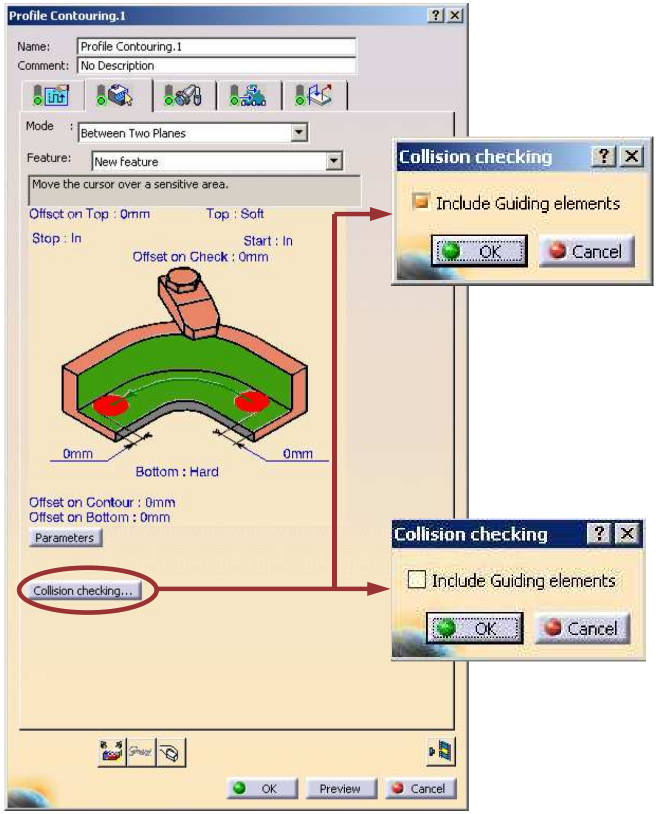
You can use Helix tool path style with machining mode as 'Between Two Planes' only.

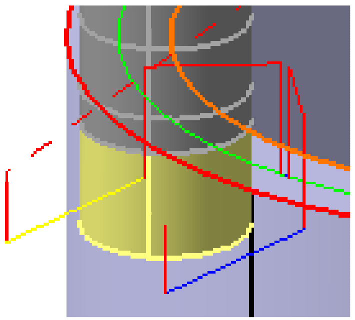
Student Notes:

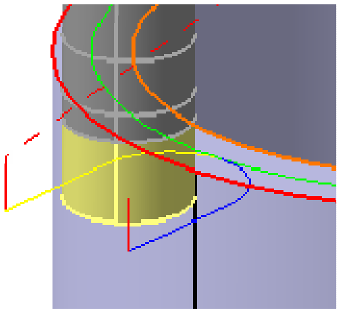
Profile Contouring Operation: Geometry (3/4)

Collision Checking:

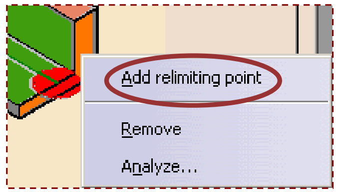
Collision checking is done during macro motion.
All guiding elements defined on the operation are taken into account during this verification.
However, in some cases, it can be useful to deactivate collision checking with the guides







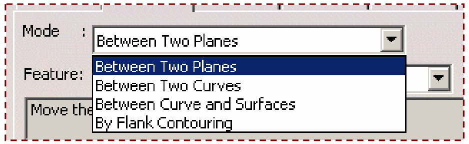
Relimiting Points



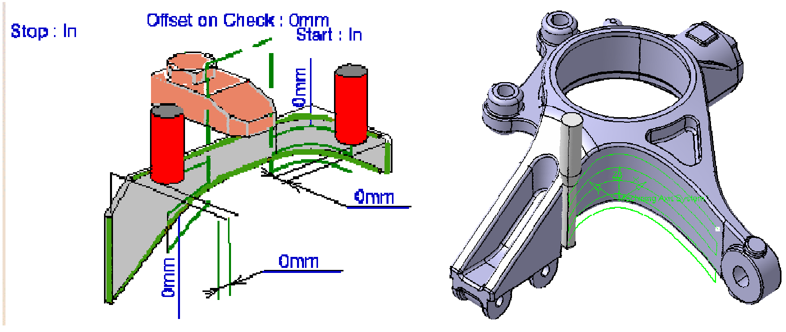
This option allows to quickly define the limit elements on the machining contour without any geometry creation.

Profile Contouring Operation: Geometry (4/4)

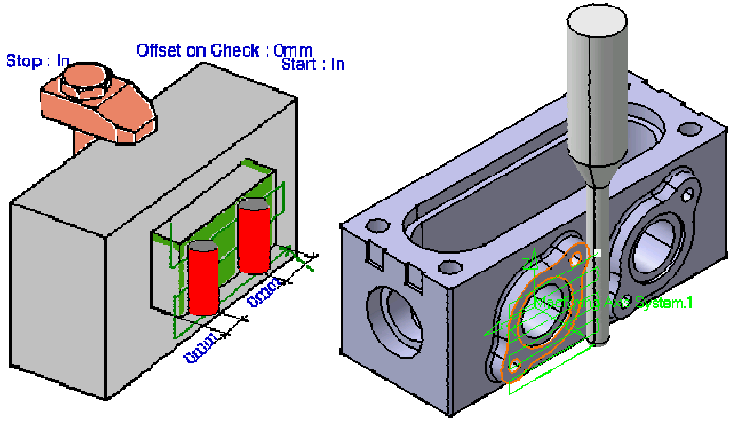
The Profile Contouring Operation Modes are:



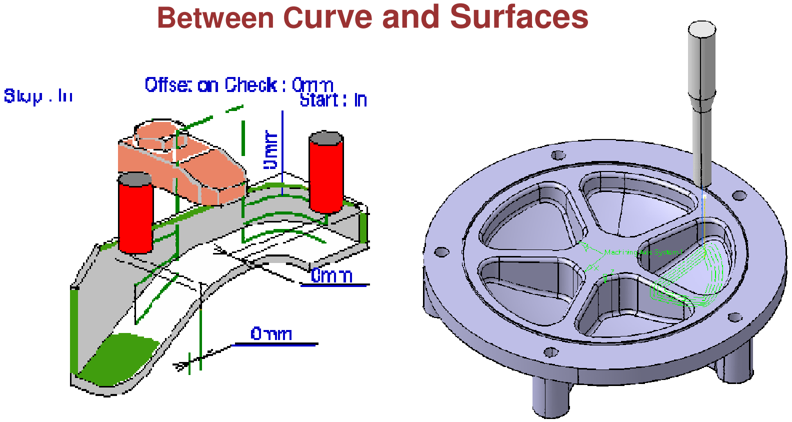
Between Two Curves

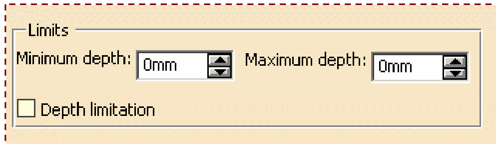


By Flank Contouring



Between Curve and Surfaces





- **Machining can be restricted to a specific zone by specifying Minimum depth and Maximum depth values.**
- **Available with 'Between Two Curves' and 'Between Curve and Surfaces'.**

Student Notes:

Student Notes:

Profile Contouring Operation: Feeds and Speeds

Feedrate Reduction in Corners:

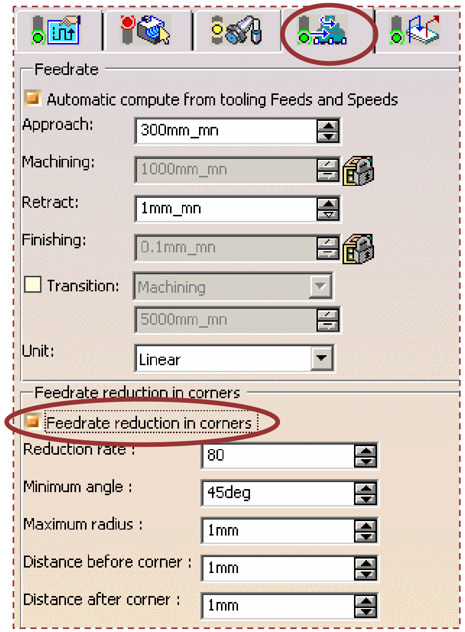
You can reduce feedrate in corners encountered along the tool path depending on values given in the Feeds and Speeds tab page

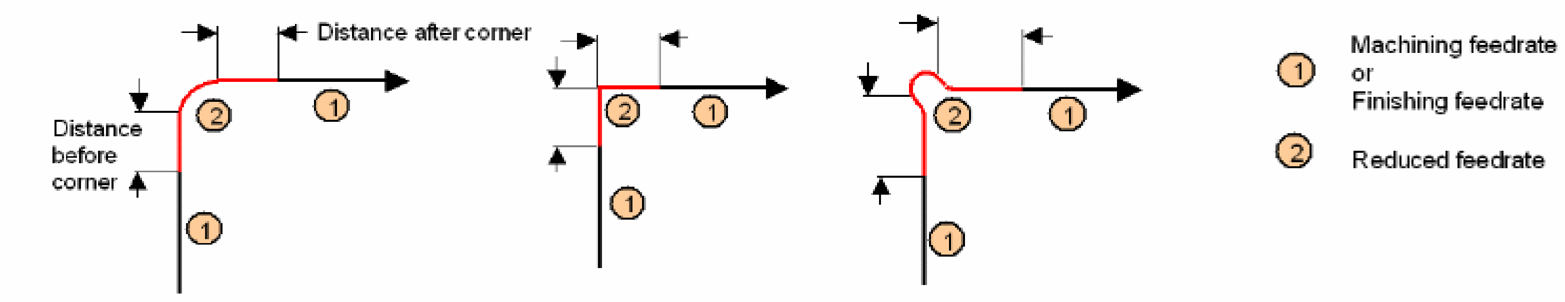
Feed reduction is applied to corners along the tool path whose radius is less than the Maximum radius value and whose arc angle is greater than the Minimum angle value.

For Pocketing, Feedrate reduction applies to inside and outside corners for machining or finishing passes.

Corners can be angled or rounded, and may include extra segments for HSM operations.

It does not apply for macros or default linking and return motions.



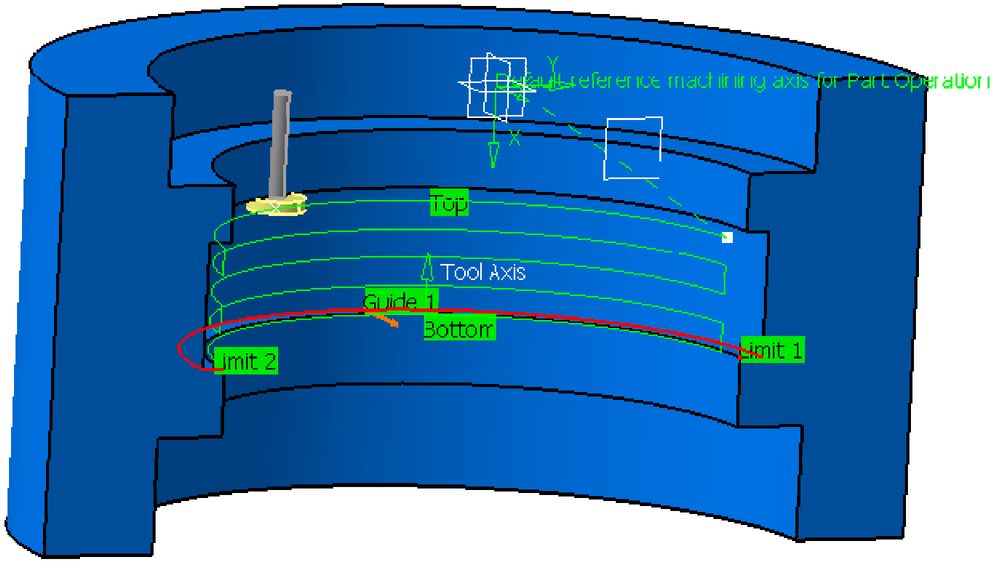


Student Notes:

Groove Milling Operation

The information in this lesson will help you to create and edit Groove Milling Operation in your Manufacturing Program.

- **Introduction**
- **Creating a Groove Milling Operation**
- **Strategy**
- **Geometry**
- **Tool**



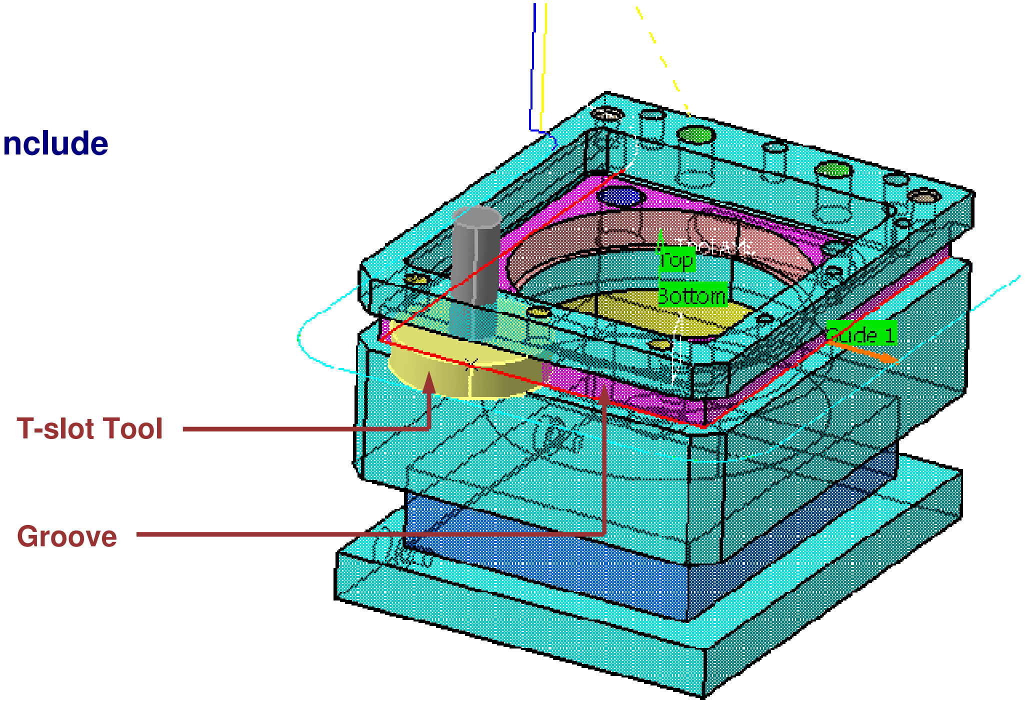
Student Notes:

Groove Milling Operation: Introduction

Groove Milling Operation allows you to machine groove area with a T-slot tool.

In Groove Milling Operation:

- **Geometry parameters include**
  - **Bottom and Top plane**
  - **Guiding elements**
- **Machining strategy parameters include**
  - **Top to bottom**
  - **Bottom to top**
  - **Middle and Alternate middle**



Student Notes:

Creating a Groove Milling Operation: General Process

1. Type the Name of the Operation. (optional because a default name is given by the system 'Type_Of_Operation.X')

2. Type the text of comment (optional)

3. Define operation parameters using the 5 tab pages

Strategy tab page

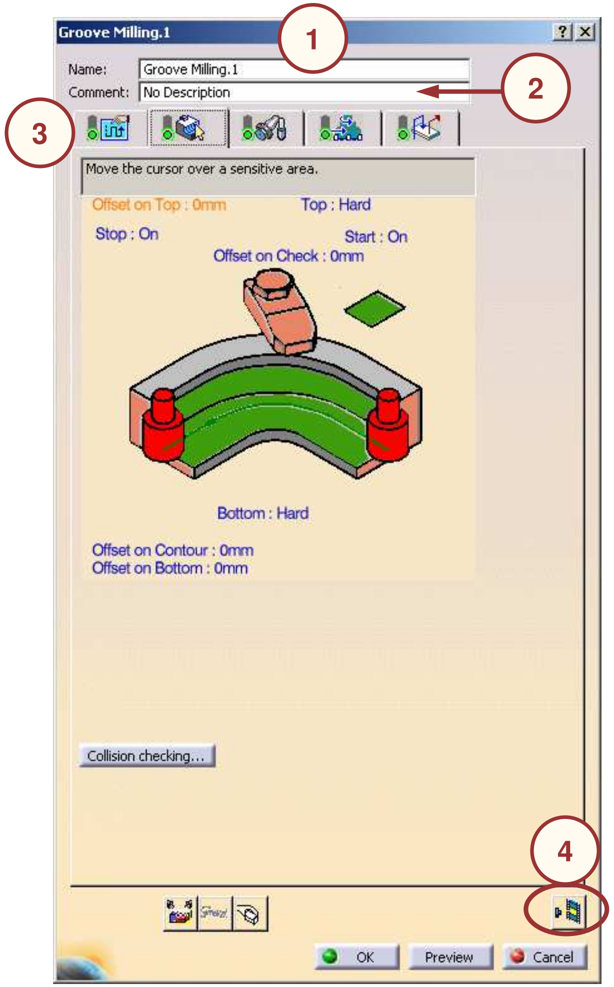
Geometry tab page

Tool tab page

Feeds & Speeds tab page

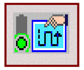
Macros tab page

4. Replay and/or Simulate the operation tool path



Student Notes:

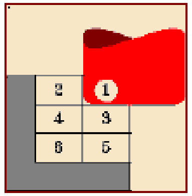
Groove Milling Operation: Strategy (1/5)

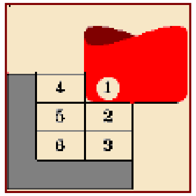


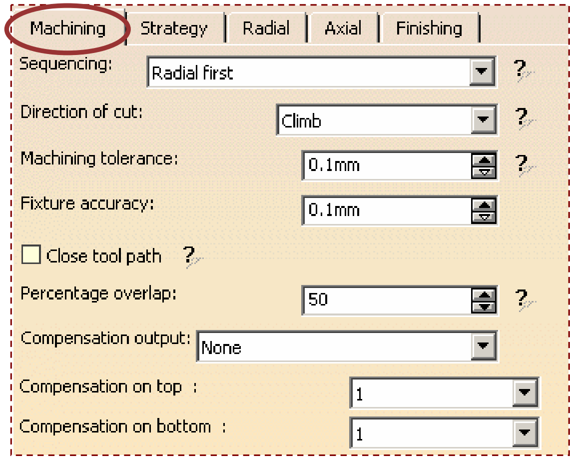
Machining Parameters:

Sequencing:

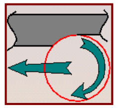
Radial first **Axial first**



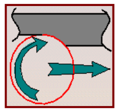




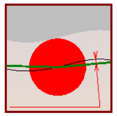
Direction of cut
Climb: The front of the advancing tool cuts into the material first



Conventional: The back of the advancing tool cuts into material first

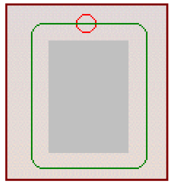


Machining tolerance:
Value of the maximum allowable distance Between the theoretical tool path and computed tool path

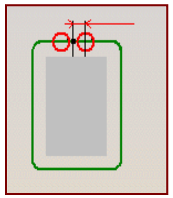


Fixture accuracy:
Local machining tolerance for fixtures

Close tool path:
Option to machine the complete contour of a closed area.



Percentage overlap:
when « close tool path » is active,this is the overlap at the end of the tool path expressed as a percentage of the tool diameter



Student Notes:

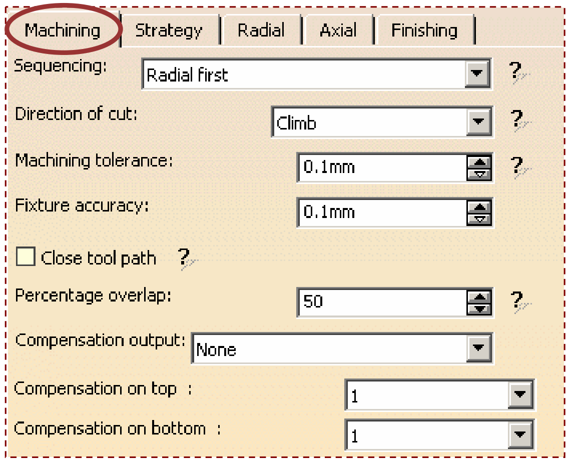
Groove Milling Operation: Strategy (2/5)

Machining Parameters:

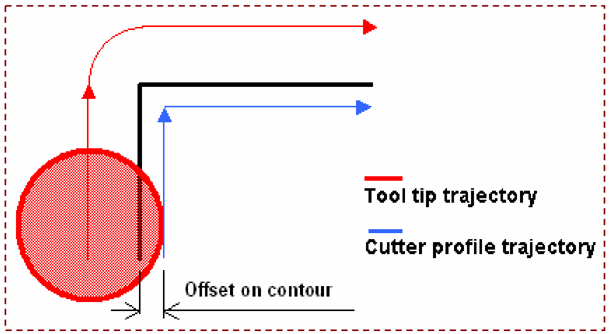
Compensation output

Generation of Cutter Compensation (CUTCOM) instructions for the groove milling tool path

- **None**
  Cutter compensation instructions are not generated in the NC data output (one can define them manually).
- **2D radial profile**
  Both the tool tip and cutter profile will be visualized during tool path replay. Cutter compensation instructions are automatically generated in the NC data output. An approach macro must be defined to allow the compensation to be applied.
- **2D radial tip**
  Tool tip will be visualized during tool path replay. Cutter compensation instructions are automatically generated in the NC data output. An approach macro must be defined to allow the compensation to be applied.



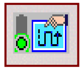
A negative Offset on contour (parameter in Geometry tab) is possible for 2D radial profile.



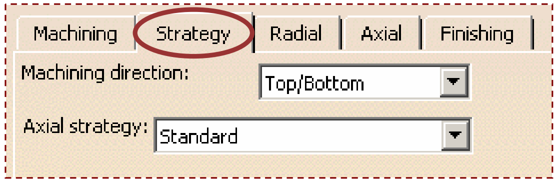
Compensation on top or bottom

Two types of compensation are available with groove milling tool. 'Number of tool compensation' must be specified on the tool that is used for the operation.

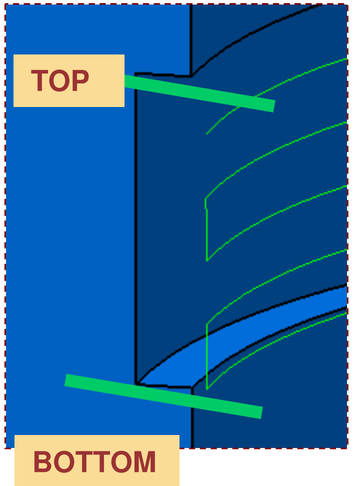
Groove Milling Operation: Strategy (3/5)

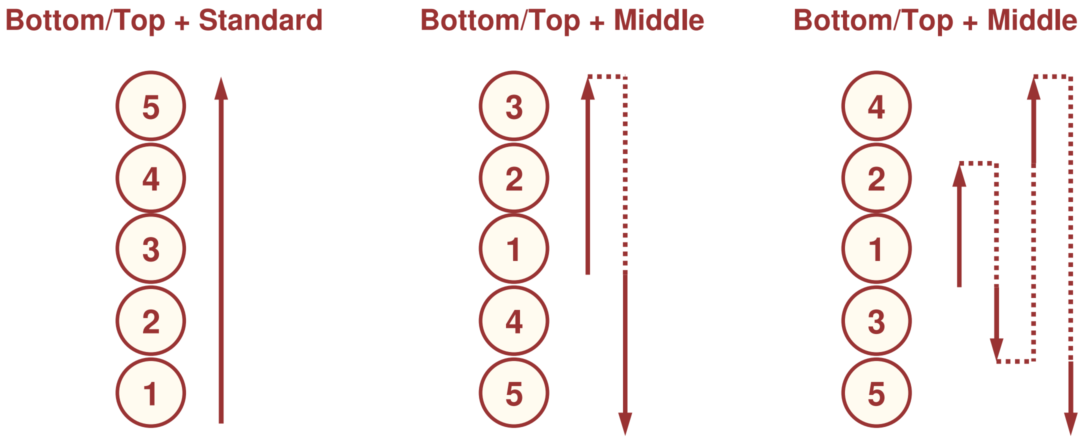


Strategy Parameters:



Machining direction and Axial strategy (in case of more than one axial level)





And reverse for Top/Bottom

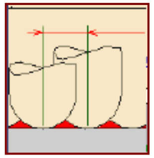
Student Notes:

Groove Milling Operation: Strategy (4/5)

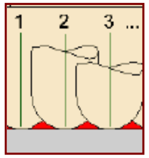
Student Notes:

Radial and Axial Parameters:

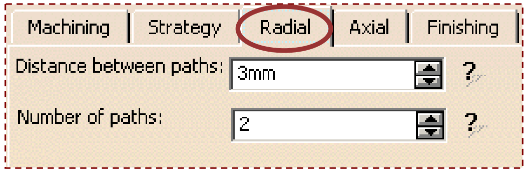
Radial Strategy



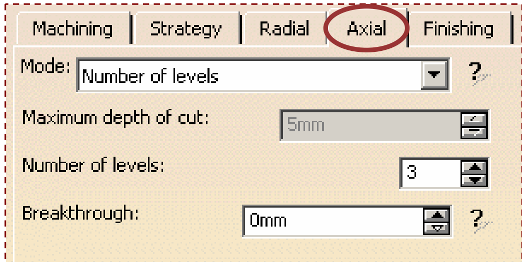
Distance between paths:
Define the distance between two radial paths



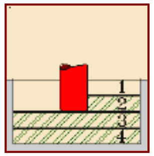
Number of paths:
Define the number of radial paths



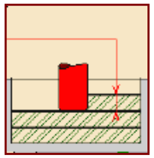
Axial Strategy



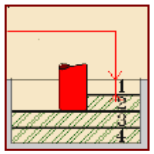
Number of levels:
Define how many levels between top and bottom planes



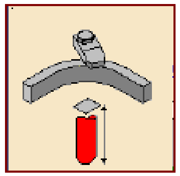
Maximum depth of cut:
Define maximum depth of cut allowed for each level



Number of levels without top
Define how many levels and maximum depth of cut, the computed tool path will not take into account the top plane.



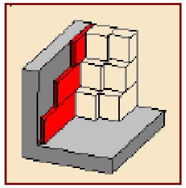
Breakthrough
Only in soft bottom, give an offset in order to specify a virtual bottom.



Student Notes:

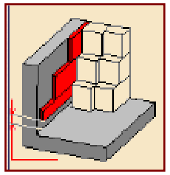
Groove Milling Operation: Strategy (5/5)

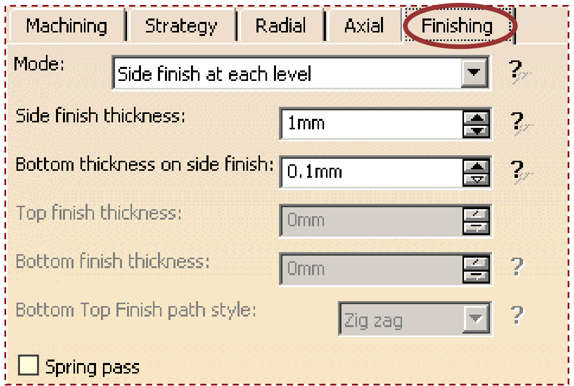
Finishing Parameters:

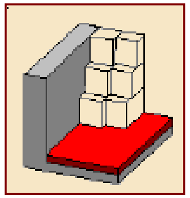


Side finish at each level:
Side finish path will occur at each level

Define Side finish thickness and a Bottom thickness on side finish (if you want to avoid to mark the bottom while doing the side finish path)



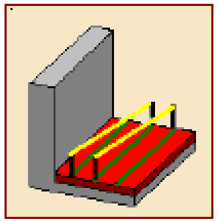


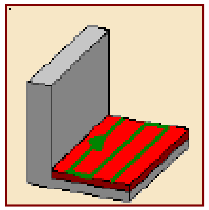


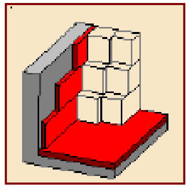
Top and/or Bottom:
Define Top finish thickness and/or Bottom finish thickness

Bottom Top Finish path style:

One way or **Zig-zag**







Side and Top and/or Bottom:
Combine the two previous solutions

Spring Pass duplicates last finish pass to compensate the spring of the tool.

Groove Milling Operation: Geometry

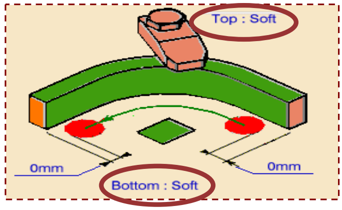
Geometry Parameters:

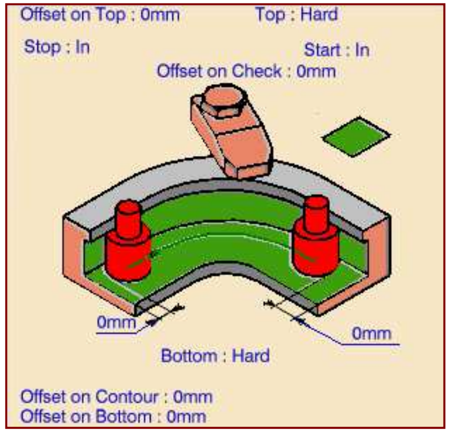
Bottom:

Plane which defines the groove bottom. May be Soft. (by default, Contour Detection is OFF)

Top:

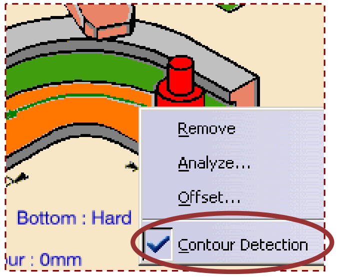
Plane which defines the top of groove; may be Soft





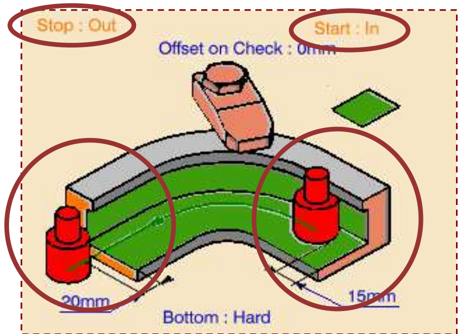
Guiding element:

Flank contour of the groove.
If Contour Detection was ON during bottom selection, boundary of selected face will be proposed.



Relimiting elements:

Use this option to manage tool start and stop position using Start and Stop positioning and offsets



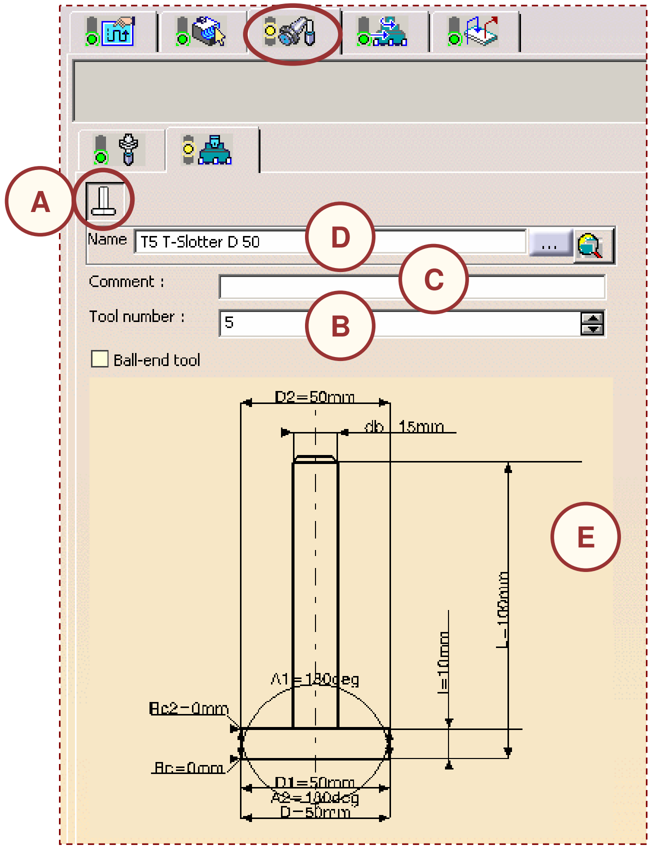
Student Notes:

Student Notes:

Groove Milling Operation: Tool

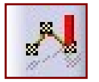
A Groove Milling operation uses only a T-Slotter.

- **A** T-Slot tool type
- **B** Type the Name of the Tool.
- **C** Type a line of comment (Optional).
- **D** Specify a tool number that already does not exist.
- **E** Use the 2D Viewer to modify the parameters of the tool. The 2D Viewer is updated with the new values.



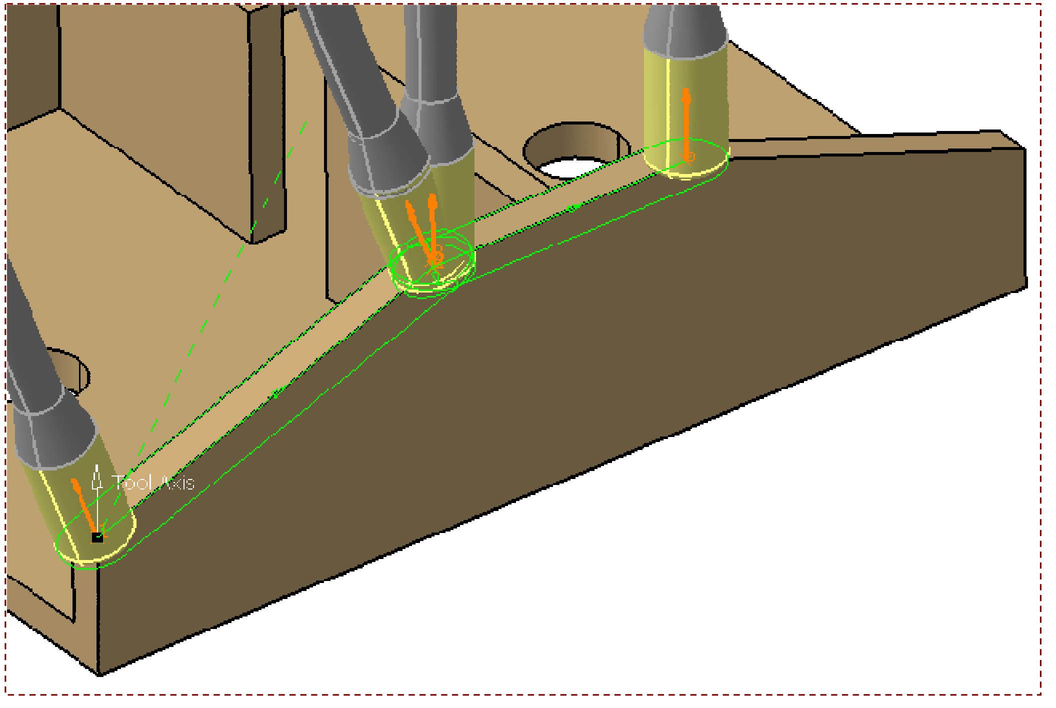
Student Notes:

Point to Point Operation



In this lesson, you will learn how to define a Point To Point Operation.

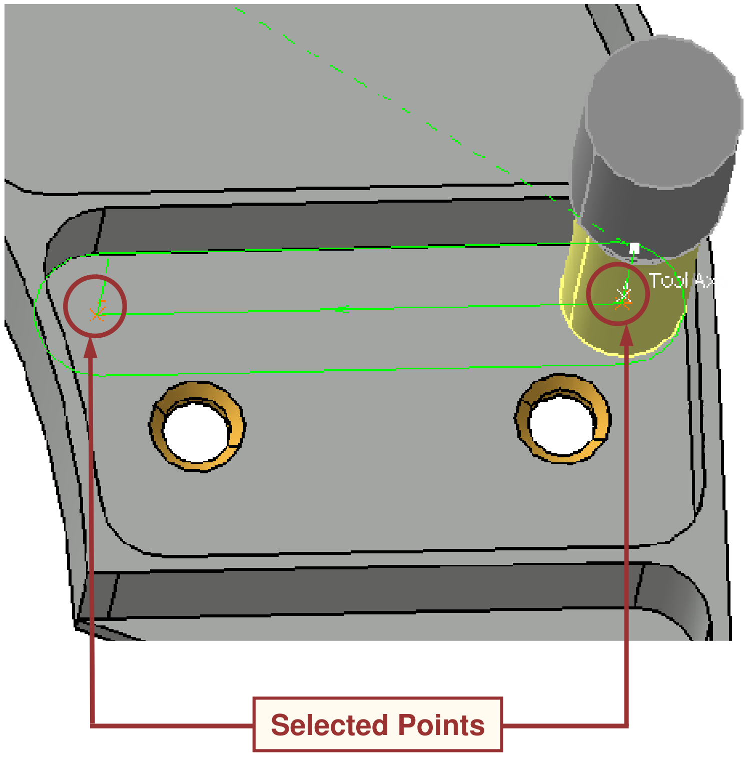
- Introduction
- Creating a Point to Point Operation
- Strategy

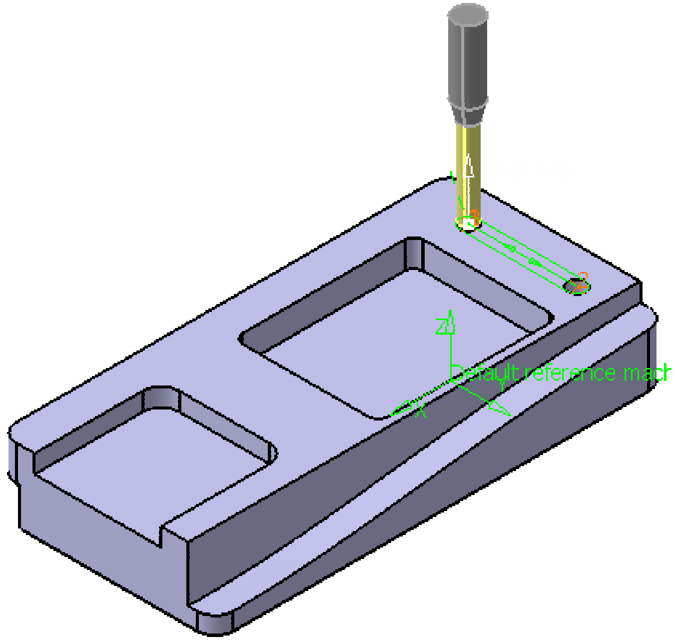


Student Notes:

Point to Point Operation: Introduction

- **A Point To Point Operation consists in moving the tool from one selected point to another selected point at a given machining feedrate.**
- **This operation allows you to define sequential motions from point to point.**





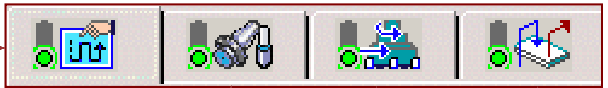
Student Notes:

Creating a Point To Point Operation: General Process

1. Type the Name of the Operation.
(Optional because a default name is given by the system 'Type_Of_Operation.X')

2. Type the text of comment (optional)

3. Define operation parameters using the 4 tab pages



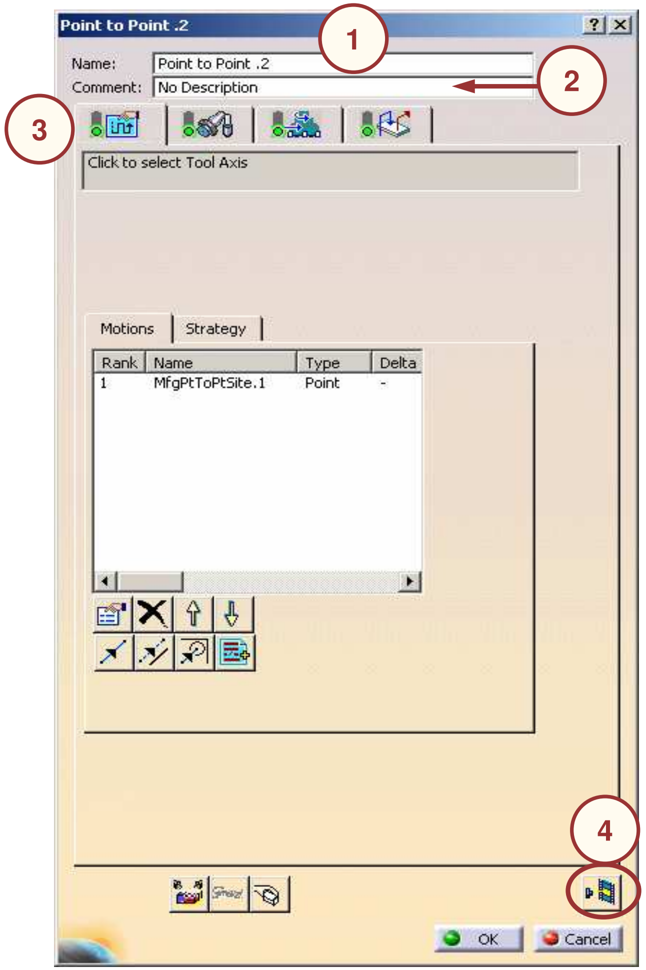
Strategy tab page

Tool tab page

Feeds & Speeds tab page

Macros tab page

4. Replay and/or Simulate the operation tool path



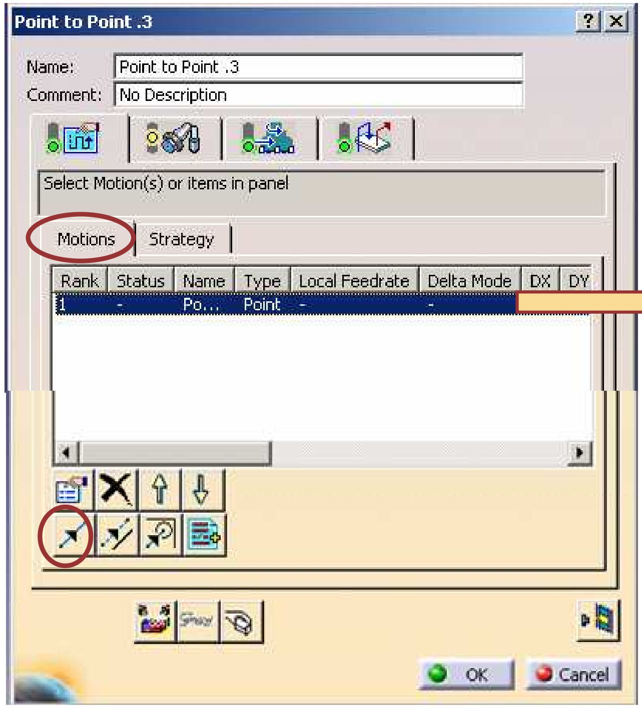
Student Notes:

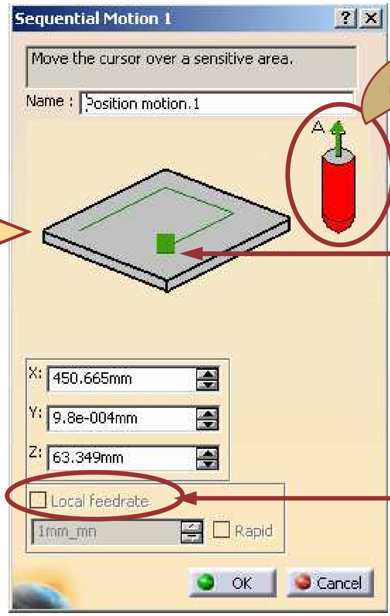
Point To Point Operation: Strategy (1/5)

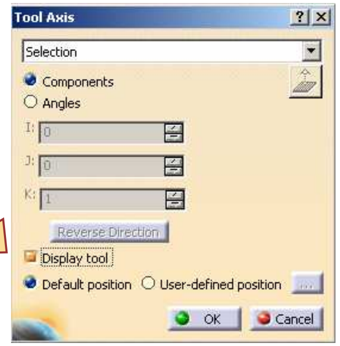
Goto Point:

A tool motion defined by the point, the tool tip has to reach.

Select Goto Point icon to specify points.
Thus you can move/delete/edit properties of selected points.







If you click on the green square, you can modify the current point directly on the geometry.

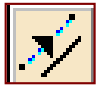
A local feedrate checkbox can be selected only if you have more than one point.

You can select different tool axes for multiple points in a single operation.

Point To Point Operation: Strategy (2/5)

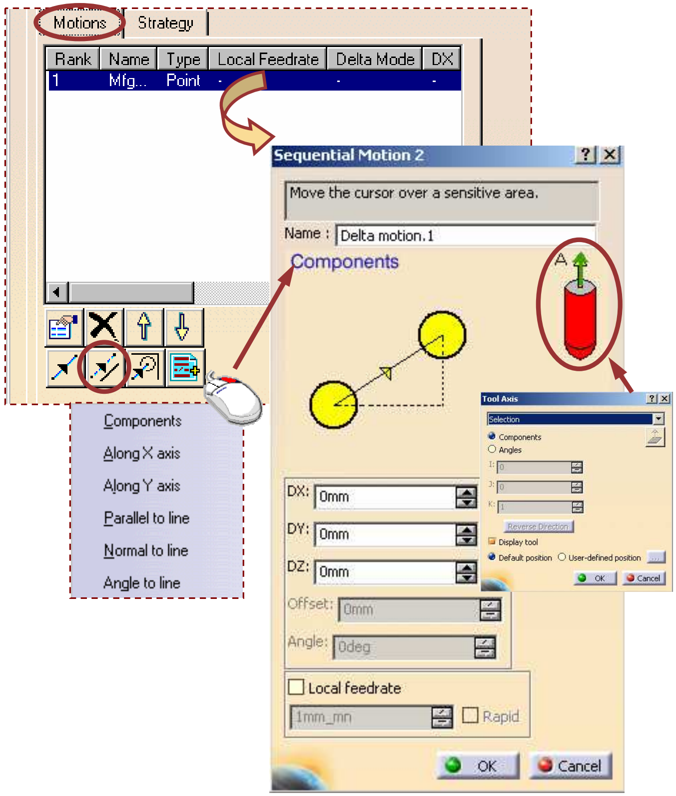
Student Notes:

Go Delta

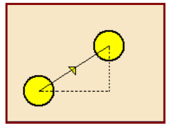


A tool motion defined by a displacement relative to a previous Point (works only if you have an existing point before), Position or GoDelta motion location. Types of Go Delta motion are as follows.

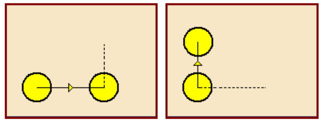
Select Goto Delta icon to specify points.



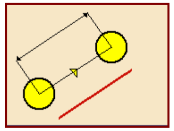
Components: relative motion defined by DX, DY, DZ displacements from previous motion location.



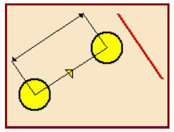
Along X or Y axis: relative motion along Y axis or X axis (current axis system) on a user specified Distance, from previous motion location.



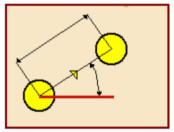
Parallel to Line: relative motion on a user specified Distance, parallel to a user selected Line, from previous motion location.



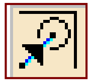
Normal to Line: relative motion on a user specified Distance, normal to a user selected Line, from previous motion location. The tool motion is done in a plane perpendicular to the tool axis.



Angle to Line: relative motion on a user specified Distance, along a line computed from user-defined Angle and Line. The tool motion is done in a plane perpendicular to the tool axis.



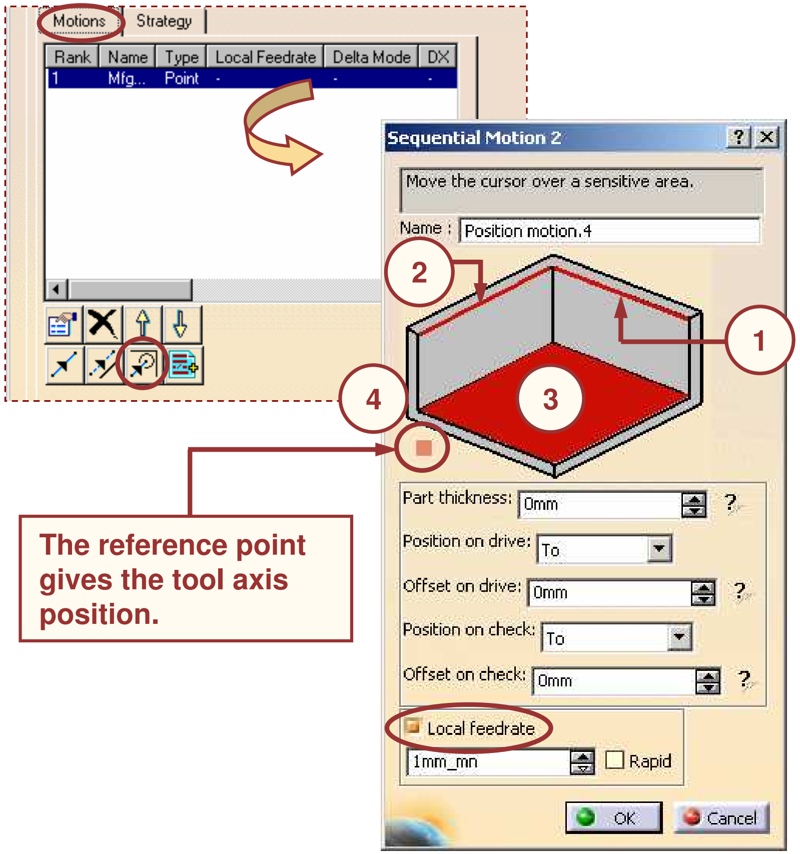
Point To Point Operation: Strategy (3/5)



Goto Position

A tool motion defined by positioning the tool in contact with a part element, a drive element and possibly a check element, while taking To / On / Past / Tgt Drive conditions into account.

Select Goto Position icon to specify points.



This Tab Page includes a Sensitive Icon Dialog box that allows the selection of:

1. **Check Element (Optional)**
2. **Drive Element**
3. **Bottom Plane**
4. **Reference Point (Optional)**

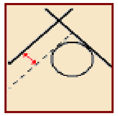
Student Notes:

Point To Point Operation: Strategy (4/5)

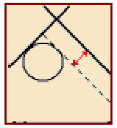
Goto Position

Offsets and Position (to,on,past) can be applied on the Drive and Check Elements.

Offset on drive

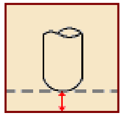


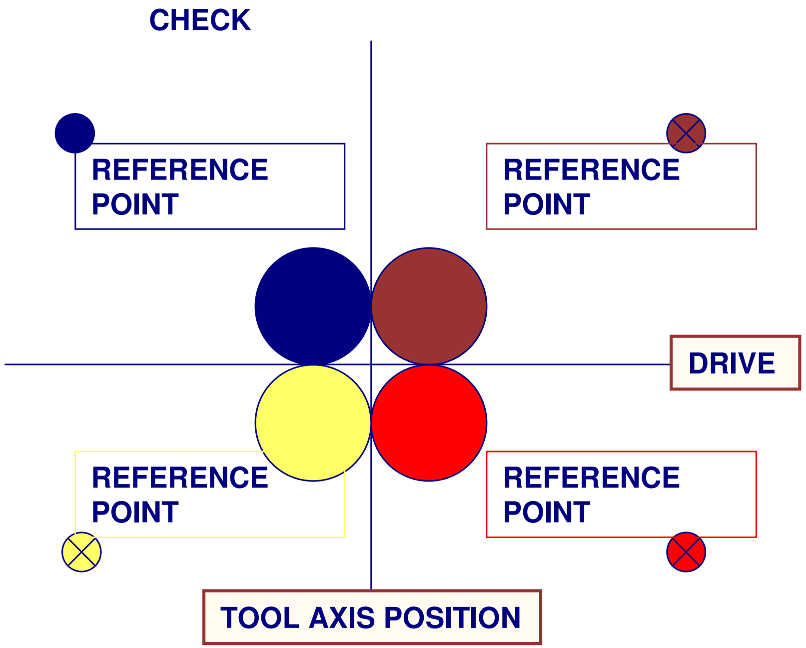
Offset on check



Part thickness can be applied.

Part thickness





The reference point gives the tool axis position.

Student Notes:

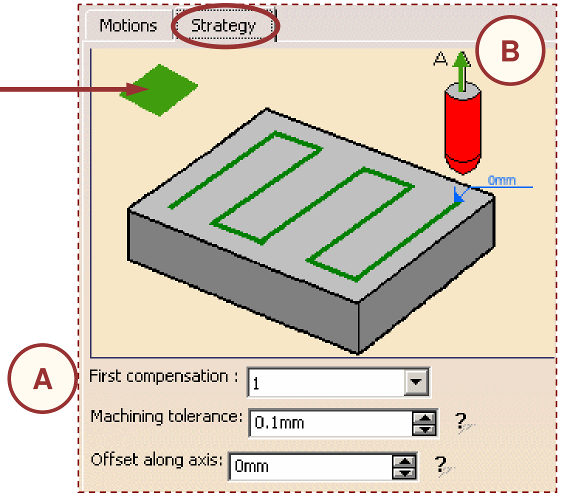
Student Notes:

Point To Point Operation: Strategy (5/5)

The Strategy tab is divided into Motions and Strategy. We have seen the Motions part, now we will see the strategy to specify machining parameters:

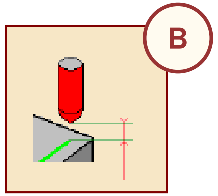
A — **Select Tool compensation number among those defined on the tool, used by the operation**

B — **The Tool Axis can be specified by selecting the tool Axis. An Offset along tool axis can be applied on all the selected points.**



By selecting a plane, all the following points that will be indicated in the 3D Window will be projected on it.

It is not necessary to create these points.

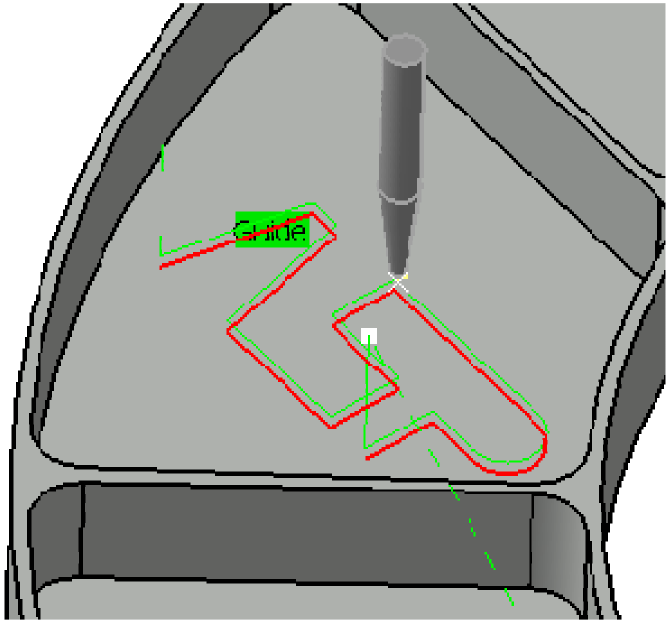


Student Notes:

Curve Following Operation

The information in this lesson will help you to define and edit Curve Following Operation in your Manufacturing Program.

- Introduction
- Creating a Curve Following Operation
- Strategy
- Geometry



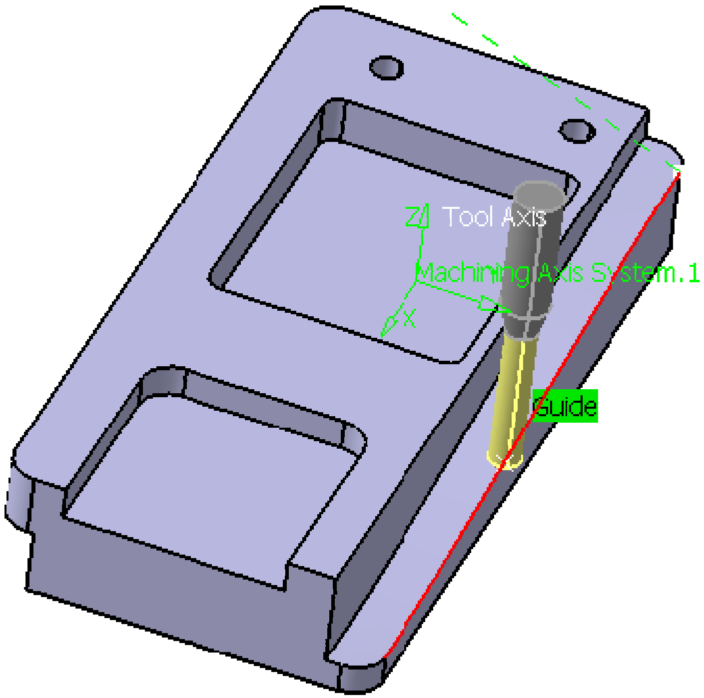
Student Notes:

Curve Following Operation: Introduction

A Curve Following Operation consists in machining a part by following a curve with the tool tip.

In a Curve Following operation:

- **The material will be removed in one or several level of cuts.**
- **The tool starts and finishes machining at the extremities of the curve.**
- **The area is machined in One-way or in Zig-zag style.**



Creating a Curve Following Operation: General Process

1. **Type the Name of the Operation. (Optional because a default name is given by the system 'Type_Of_Operation.X')**

2. **Type the text of comment (Optional)**

3. **Define operation parameters using the 5 tab pages**

Strategy tab page

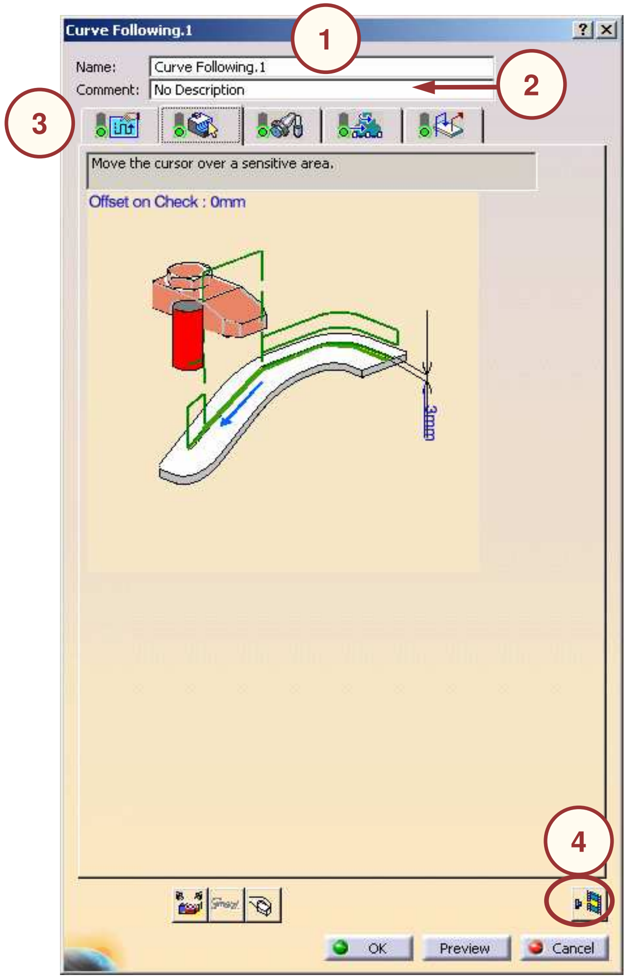
Geometry tab page

Tool tab page

Feeds & Speeds tab page

Macros tab page

4. **Replay and / or Simulate the operation tool path**



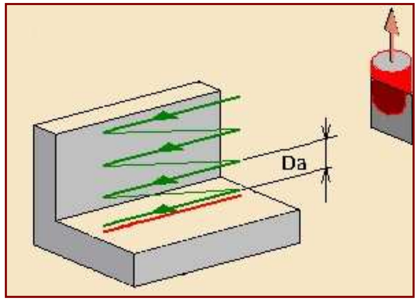
Student Notes:

Student Notes:

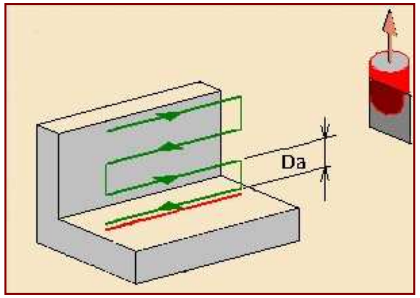
Curve Following Operation: Strategy (1/2)

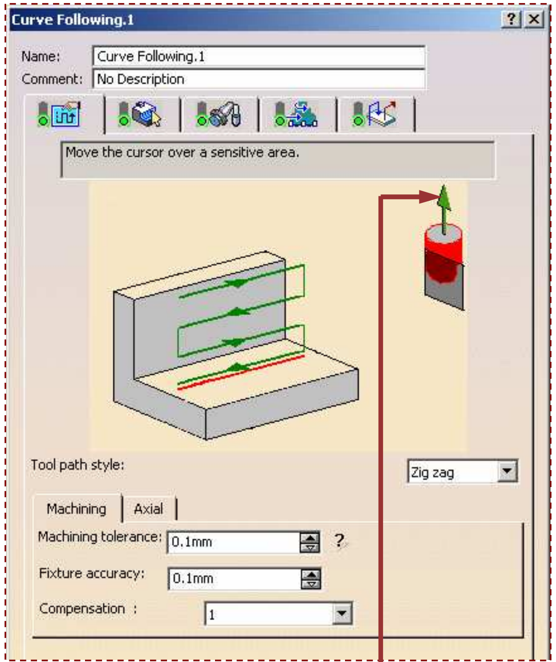
Curve following Tool path style:

One Way:
The tool machines always in the same direction.



Zig - zag:
The tool alternatively machines in one direction then in opposite direction.



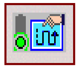


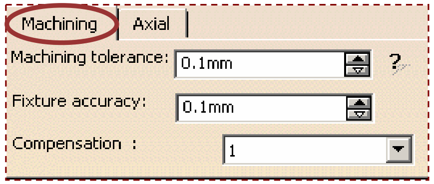
To change the tool axis orientation select the Axis representation

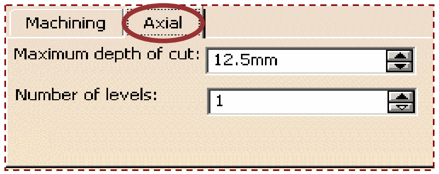
Compensation:
Define the tool corrector identifier to be used in the operation.
The corrector type, corrector identifier and corrector number must be defined on the tool.

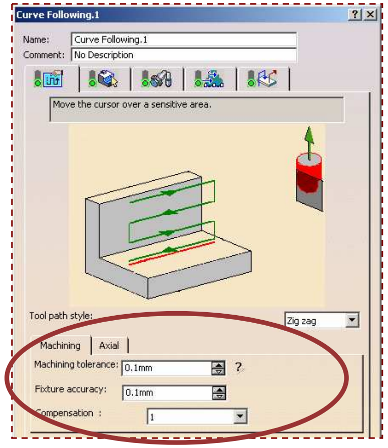
Student Notes:

Curve Following Operation: Strategy (2/2)



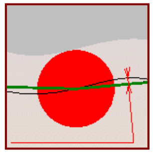






Machining Strategy Parameters:

Machining Tolerance
Value of the maximum allowable distance between theoretical tool path and computed tool path

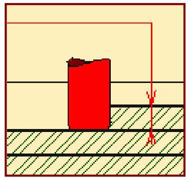


Fixture Accuracy
Local machining tolerance for fixtures

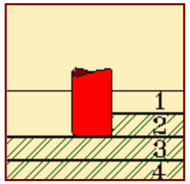
Compensation
Number of the tool compensation. It must be a number available on the tool used for the operation.

Axial Strategy Parameters:

Max depth of cut
Defines the maximum distance between two levels



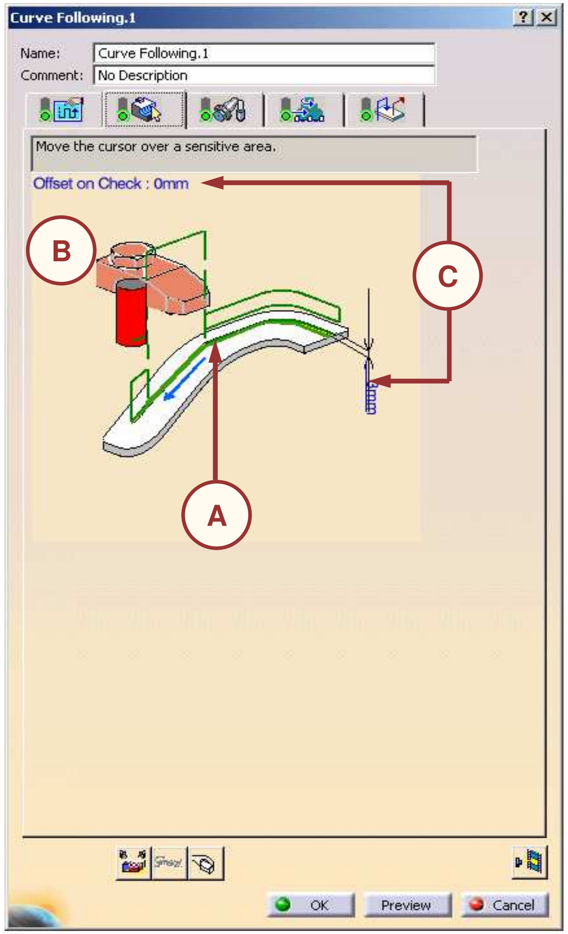
Number of levels
Define number of levels from the top to the bottom



Curve Following Operation: Geometry

This Tab Page includes a sensitive icon Dialog Box that allows the selection of:

- **A** Guiding Element
- **B** Check Elements (Optional)
- **C** Offsets (axial) can be applied on Check Elements and Driving Curve



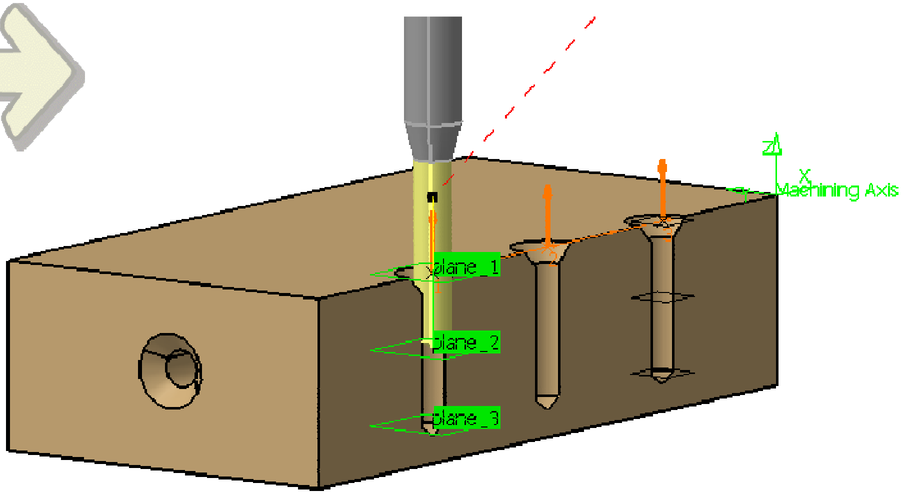
Student Notes:

Student Notes:

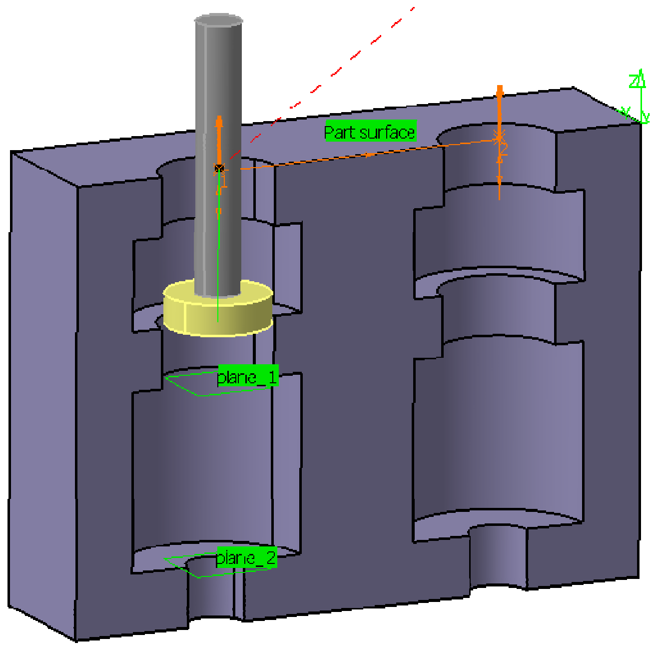
Sequential Axial and Groove Operations

In this lesson, you will become familiar with Sequential Axial and Sequential Groove Operations.

- Introduction
- Creating a Sequential Operation
- Strategy
- Geometry
- Tools
- Feeds and Speeds
- Macro



Sequential Axial Operation



Sequential Groove Operation

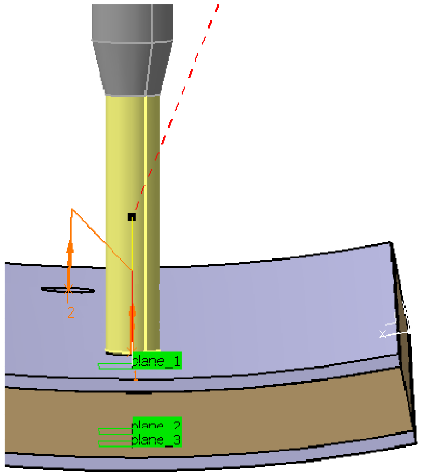
Student Notes:

Sequential Operations: Introduction

A Sequential Operation is used to machine holes or grooves with a specific machining methodology. This methodology consists of tool motions such as go to a specific position, spindle speed management, delay and circular motion.

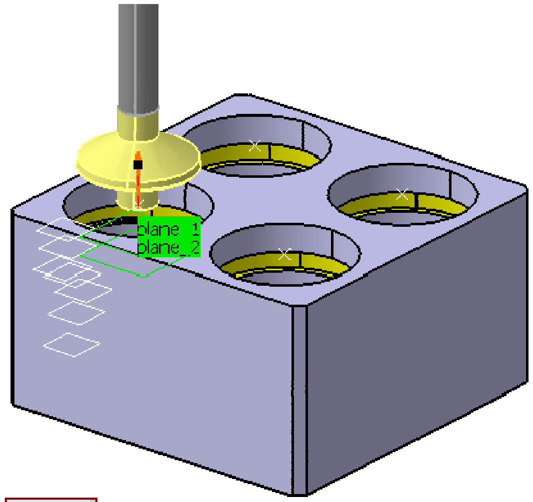
The sequential operation is used to define various tool motions in a single machining operation. This will reduce the list of operations in a manufacturing program and modifications can be easily managed.

The sequential operation can be used effectively to machine parts of composite materials where you can use different feedrates, spindle speeds, PP Words according to the type of material.



Composite material

Sequential Axial Operation



Sequential Groove Operation

Tool motions at different planes, with different speeds, feeds, and PP words.

Student Notes:

Creating a Sequential Operation: General Process

1 Type the Name of the Operation.
(optional because a default name is given by the system 'Type_Of_Operation.X')

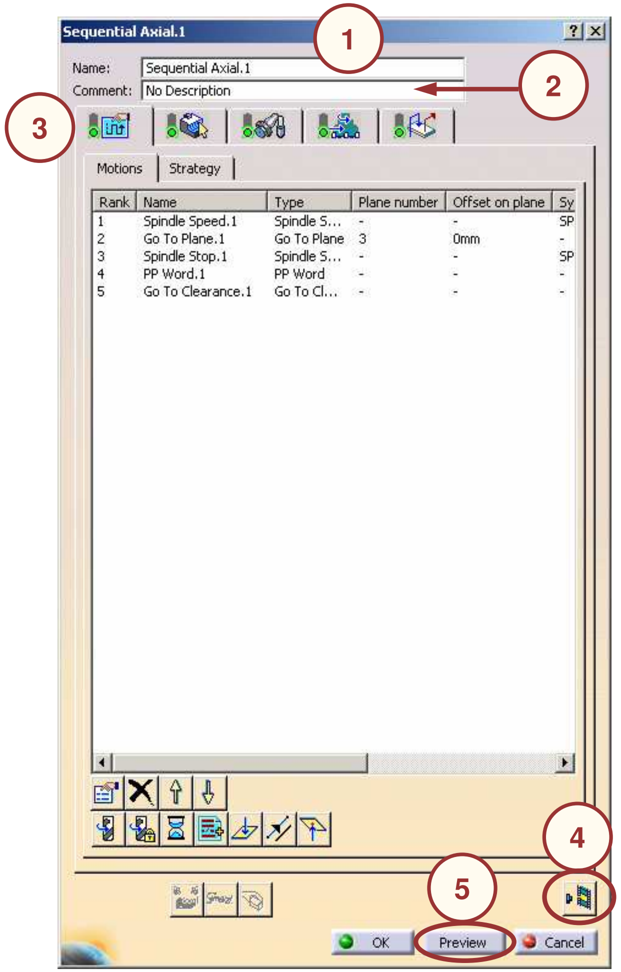
2 Type the text of comment (optional)

3 Define operation parameters using the 5 tab pages

- Strategy tab page
- Geometry tab page
- Tool tab page
- Feeds & Speeds tab page
- Macros tab page

4 Replay and/or Simulate the operation tool path

5 Before replaying or creating the operation, "Preview" checks that all parameters are coherent



Sequential Axial Operation: Strategy (1/9)

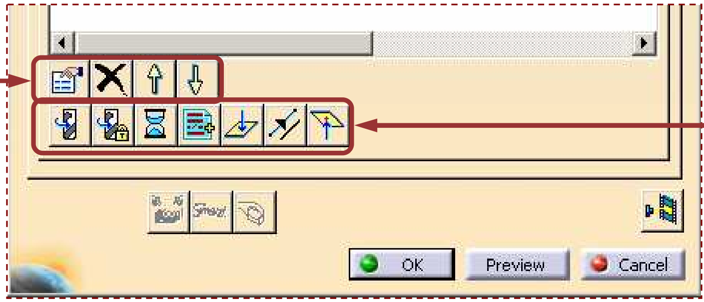
Student Notes:

You can define two sub-tab pages for Strategy tab page as Motions and Strategy

Motions: It allows you to define elementary motions to be applied on each machining hole or groove.

You can customize the column list using Column Order and Column Filter

You can Edit, Delete and Move (up or down) a sequential motion in the list.



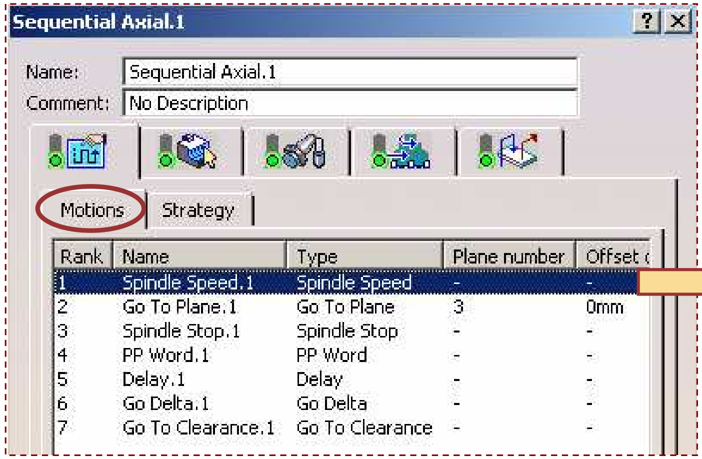
You can define Sequential motion for a operation

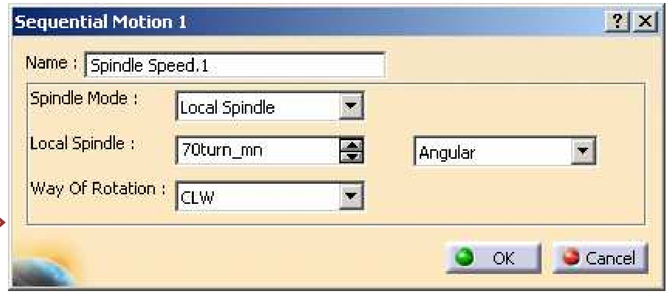
Student Notes:

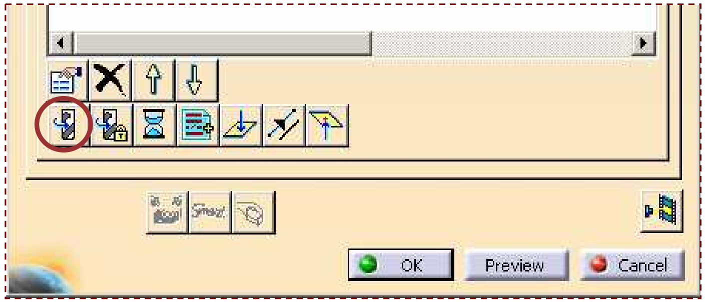
Sequential Axial Operation: Strategy (2/9)

Spindle Speed:

You can define a tool motion by Spindle Speed.







Two spindle modes:

- **Local Spindle: The tool motion follows local spindle values.**
- **Machining: The tool motion follows spindle parameters specified in Feeds and Speeds tab. You cannot specify spindle values using this option.**

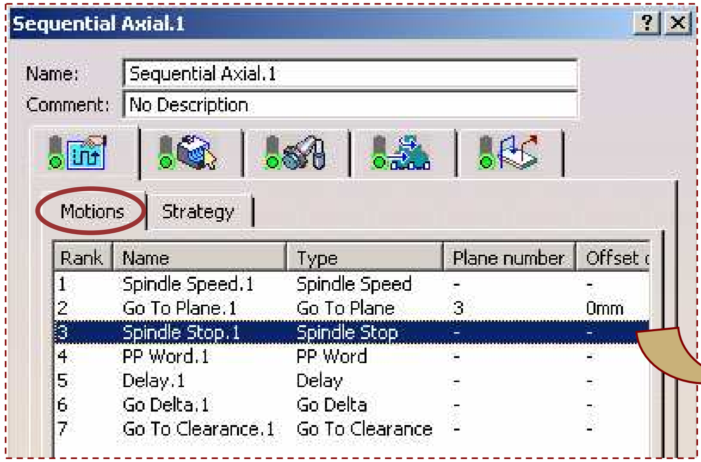
The spindle rotation can be clockwise or counter clockwise in Local Spindle mode only.

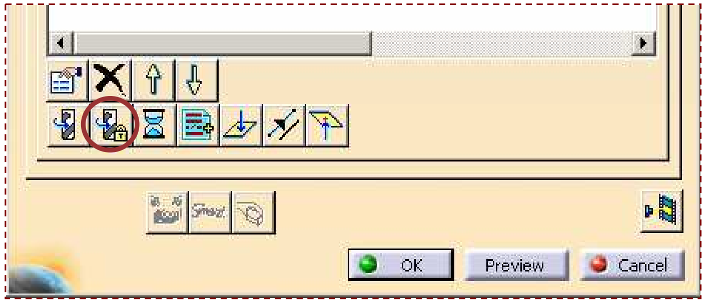
Student Notes:

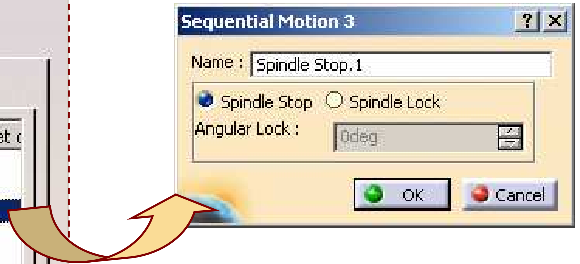
Sequential Axial Operation: Strategy (3/9)

Spindle Stop:

You can define a tool motion by Spindle Stop or Lock.







- **Spindle Stop: 'NC_SPINDL_STOP' NC Command is generated in output file**
- **Spindle Lock: 'NC_SPINDL_LOCK' NC command is generated in output file**

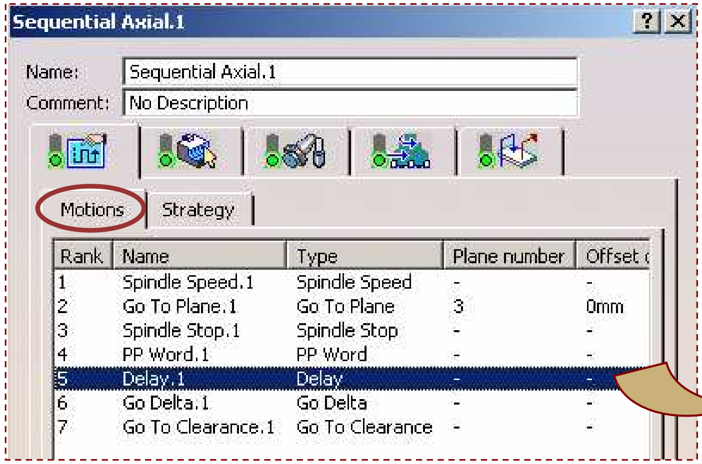
The Angular Lock angle can be entered for Spindle Lock

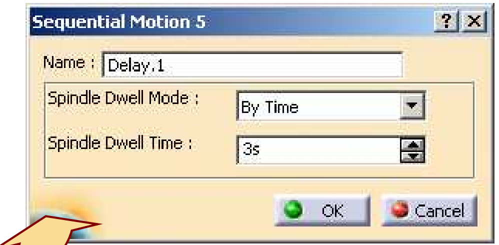
Student Notes:

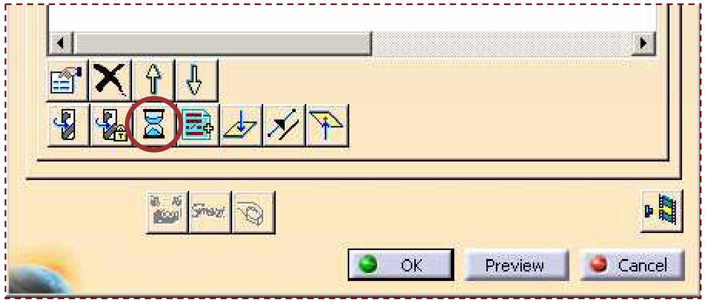
Sequential Axial Operation: Strategy (4/9)

Delay:

You can define a tool motion by a delay (in spindle revolutions or time).







Two spindle Dwell modes:

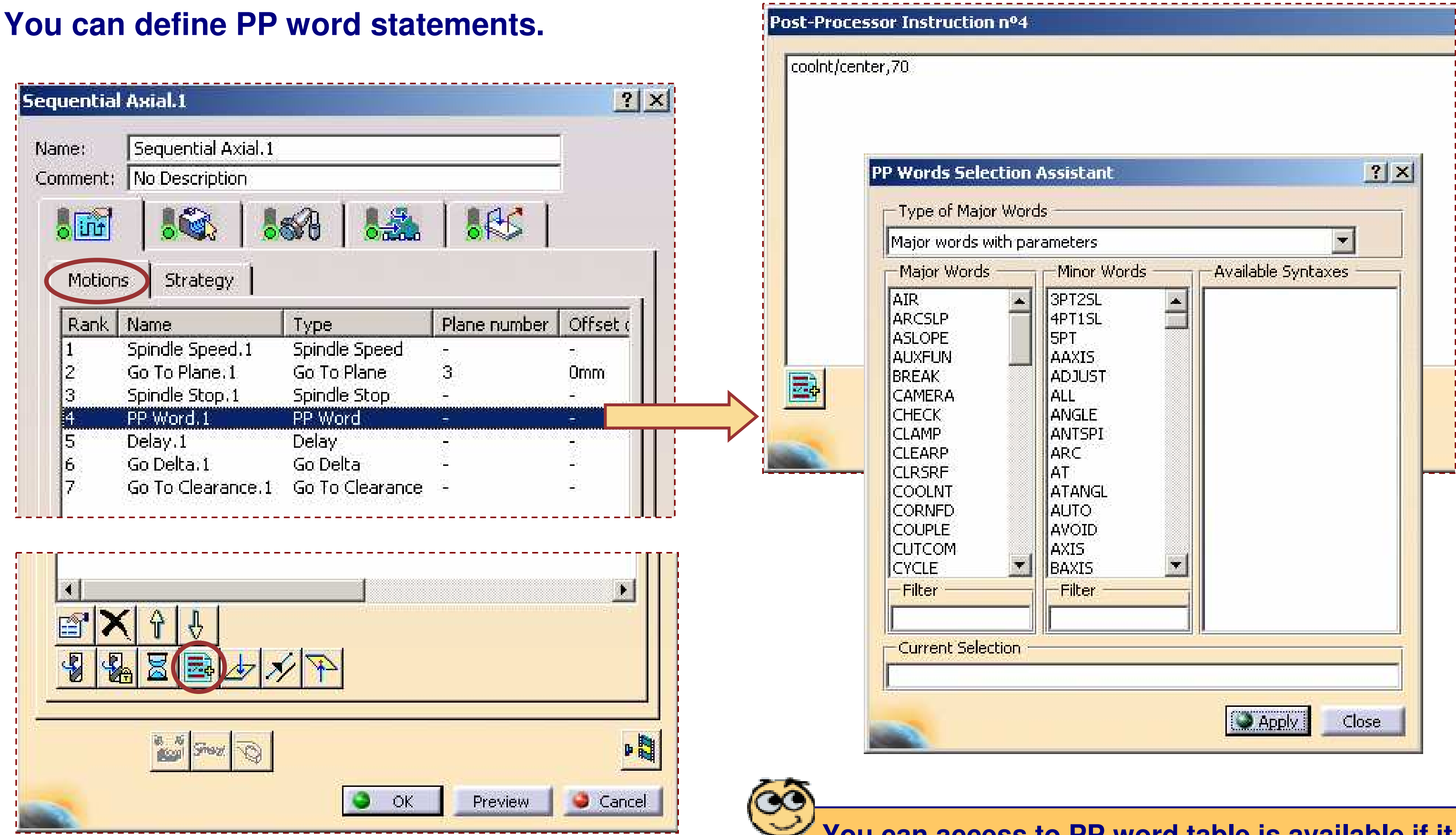
- **By Time (in sec)**
- **By Revolution (in number)**

Student Notes:

Sequential Axial Operation: Strategy (5/9)

PP Word:

You can define PP word statements.

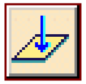


You can access to PP word table is available if it is defined on the machine of Part Operation.

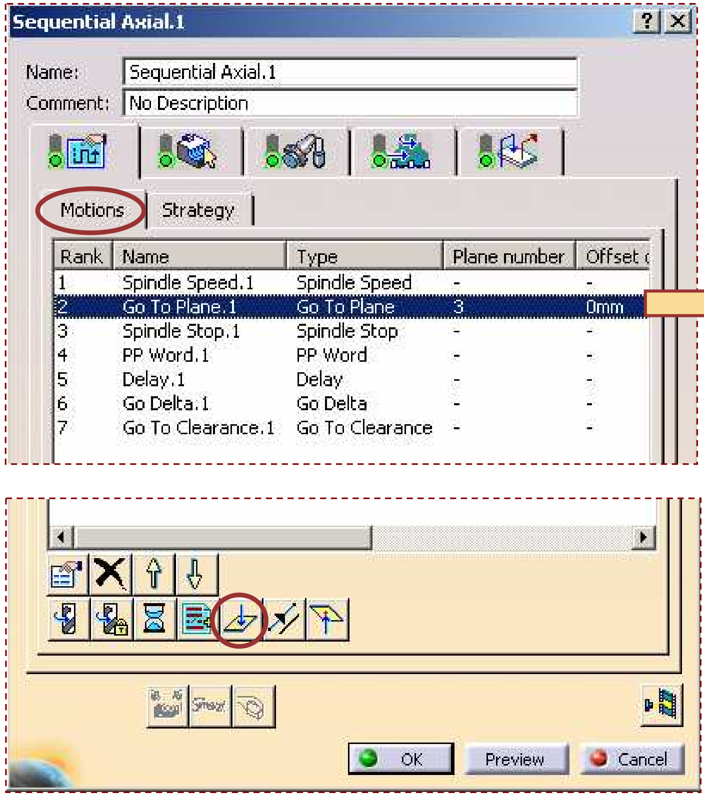
Sequential Axial Operation: Strategy (6/9)

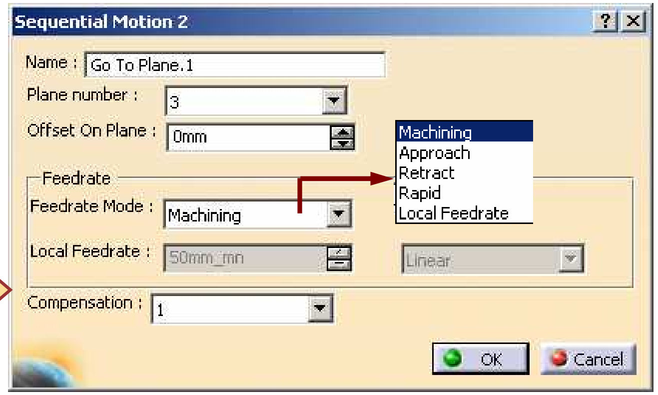
Student Notes:

Go To Plane:



You can define a tool motion by an axial motion to a plane defined and numbered in Geometry tab. The motion is done normal to the plane.





- **Offset On Plane: This offset is added to the offset that can be set on the geometrical element selected in Geometry tab.**
- **Feedrate Mode: This mode allows definition of Machining, Approach, Retract, Rapid or Local Feedrate.**
- **Compensation: The option allows you to define the Compensation point for this motion. The compensation is activated at the start of the motion.**

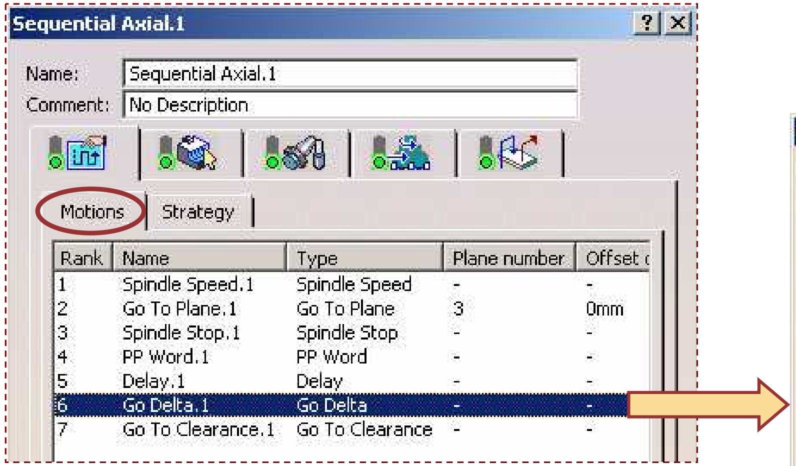
Offset is positive along the tool axis direction and negative on the opposite direction.

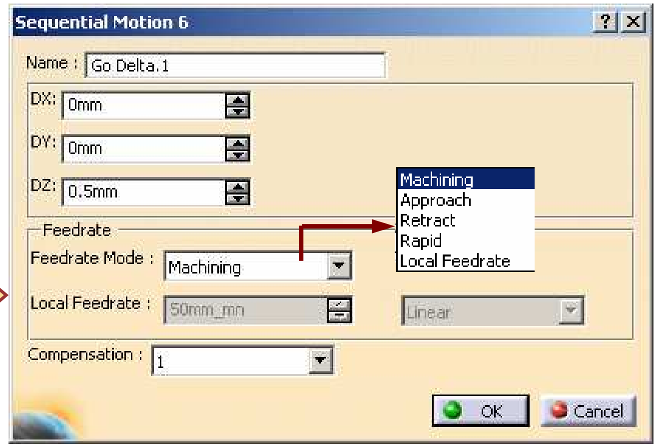
Student Notes:

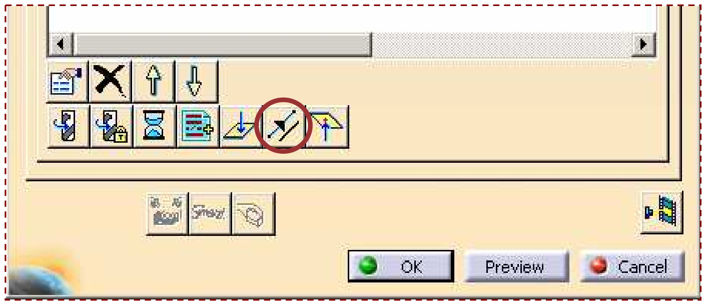
Sequential Axial Operation: Strategy (7/9)

Go Delta:

You can define a tool motion by a displacement specified by DX, DY, DZ values. Positive DZ value is defined along the machining hole axis.







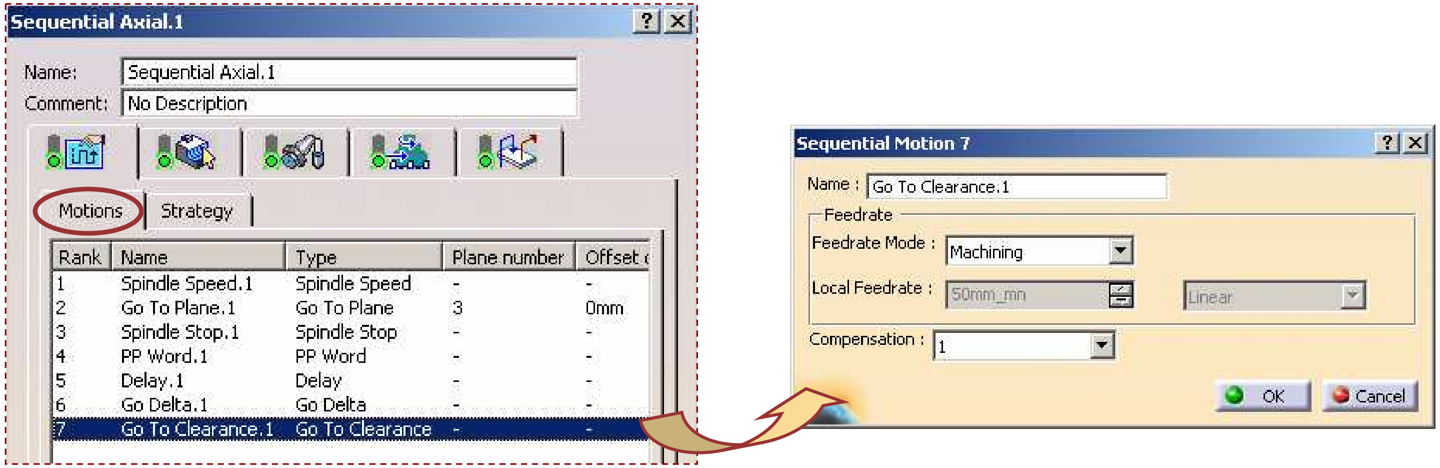
The Feedrate and Compensation definitions are same as for 'Go To Plane' Sequential Motion.

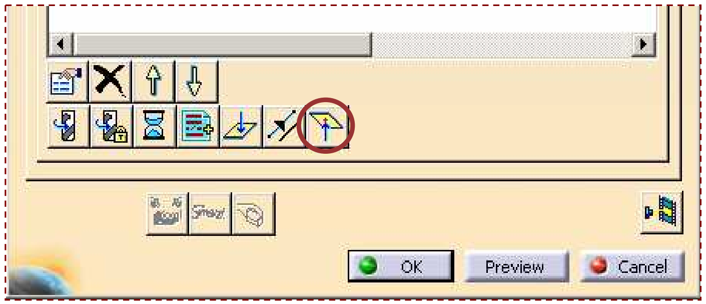
Sequential Axial Operation: Strategy (8/9)

Student Notes:

Go To Clearance:

You can define a tool motion by an axial motion up to clearance plane. The tool tip will reach the plane defined by the approach clearance displayed on the Strategy tab page.





The Feedrate and Compensation definitions are same as for 'Go To Plane' Sequential Motion.

Default behavior: If no 'Go To clearance' motion is defined in the motion list, an automatic motion is done from the last position reached by the tool motion (last sequential motion) to the clearance plane. This automatic motion is done at rapid feedrate.

Sequential Axial Operation: Strategy (9/9)

Student Notes:

You will now see the second sub-tab page.

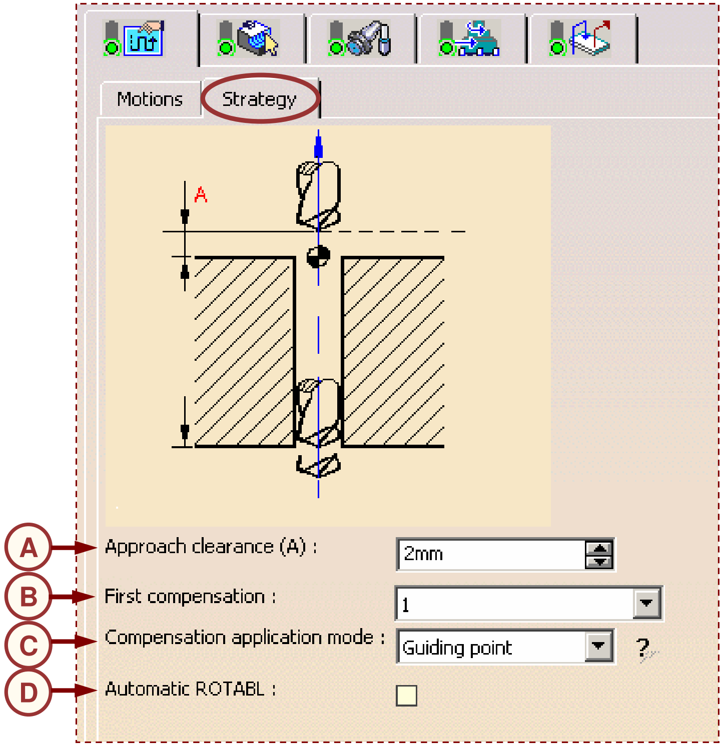
Strategy: It allows you to define machining parameters.

A. **Approach clearance: It defines the safety distance along the tool axis for approaching the hole reference.**

B. **First compensation: It specifies the tool compensation point used at the start of the toolpath for each machining hole.**

C. **Compensation application mode: It defines how the tool compensation is used to compute tool motion (guiding point) or only used at output file generation (output point).**

D. **Automatic ROTABL: It allows the generation of rotation motions between drilling points that have different tool axes. This capability works with a 3-axis milling machine with rotary table when ROTABL/ output is requested.**



Student Notes:

Sequential Axial Operation: Geometry (1/2)

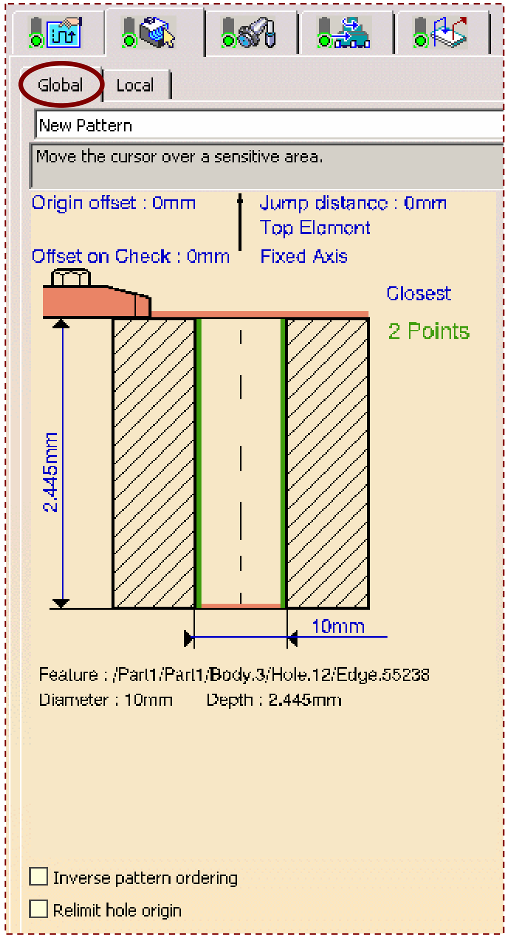
- Geometry selection is dedicated to hole machining.
- You can define planes (and offset) or depth values to manage axial toolpath between planes.
- You can select several planes (up to 5 planes at a time) and use them for the tool motions.

Global:

The geometry (hole to machine) is managed in 'Global' tab page.

Diameter and Depth are initialized from selection (same as Drilling operation). They are not used for toolpath computation but are displayed as information and can be used by f(x) formula.

You have the following functionalities to define a Sequential Axial Operation geometry:
Check element selection, Offset on check, Top element selection, Top element/Projection, Origin offset, Jump distance, Machining points to select (selection and management of machining points), ordering capability (Closest, Manual, By Band), Machining Pattern selection, and so on.



> **The depth is defined for the first machining hole and the same depth value will be used to machine all the machining holes.**

Student Notes:

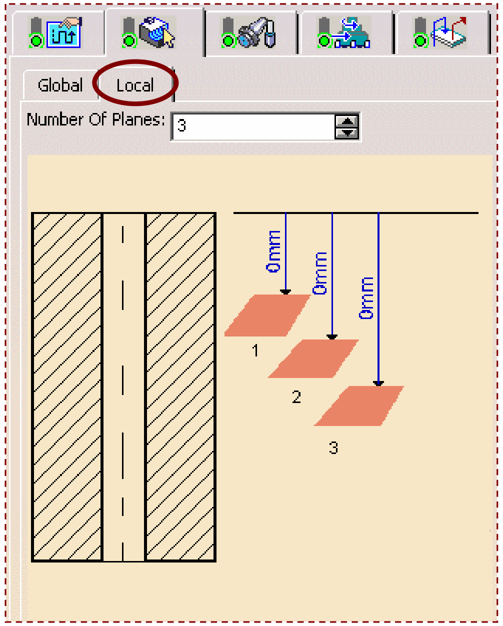
Sequential Axial Operation: Geometry (2/2)

Local:

The additional geometry (planes or depth and offsets) is managed in 'Local' tab page.

You can define a Number of machining planes (maximum 5 at a time) and Depth (number of depths depends on number of machining planes). Depth can be defined by value or by geometrical selection (plane, planar surface, planar edge, or point).

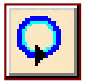
You can define Offset on the selected element (machining plane) by right-clicking on it.



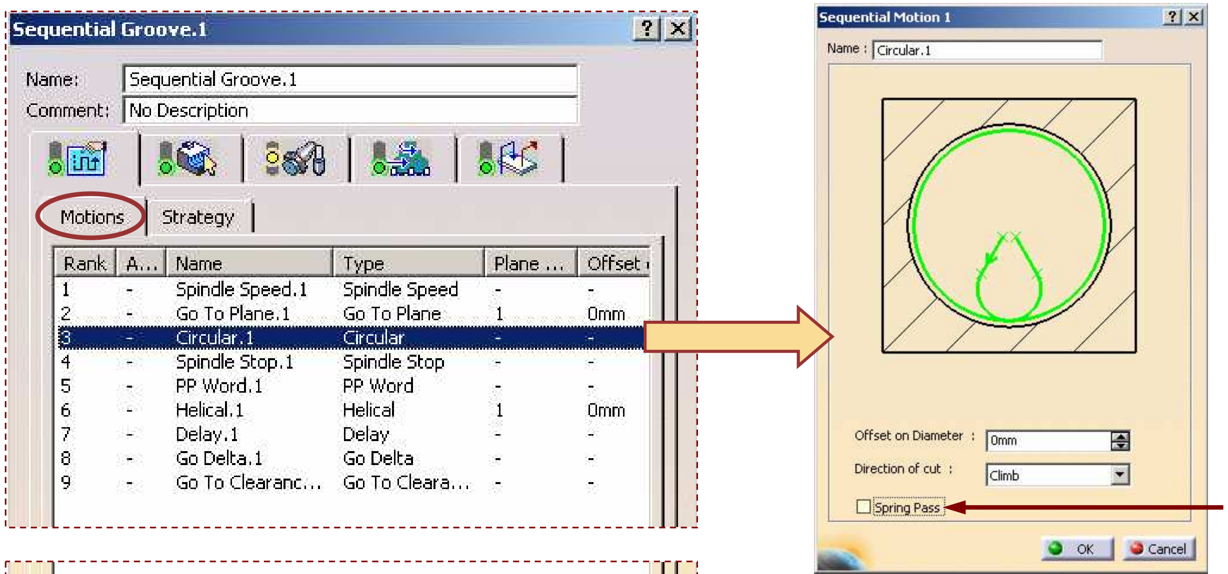
Student Notes:

Sequential Groove Operation: Strategy (1/2)

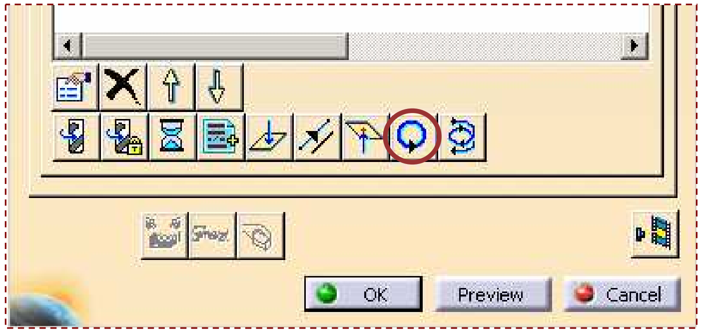
Circular:



You can define a tool motion by approach, retract and complete circular motions in one dialog box.



An optional circular motion before retract motion.



Toolpath generation: Approach motions, Circular motion (including offset) and Retract motions.

Approach and Retract motions can be deactivated, but Circular motion cannot be deactivated.

You can set:

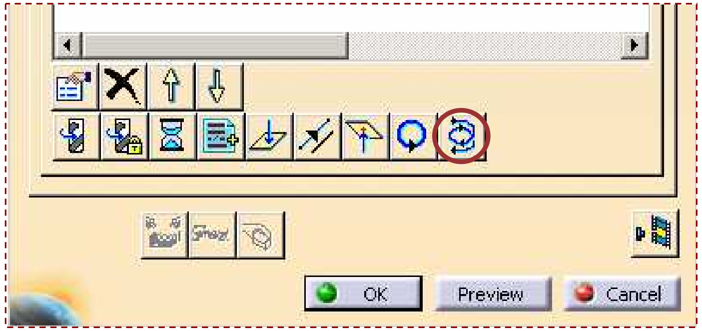
- **Feedrate type, or local value (including feedrate unit) on each motion.**
- **Circular approach and retract motion parameters.**

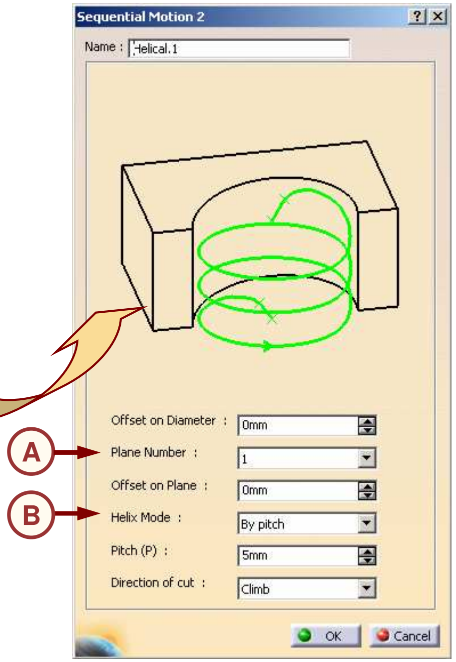
Sequential Groove Operation: Strategy (2/2)

Student Notes:

Helical:

You can define a tool motion by approach, retract and complete helical motions in one dialog box.





A. **Plane Number:** The plane to reach (plane 1 or plane 2 of the level).

B. **Helix Mode (by angle/by pitch):** Enables display of angle or pitch value.

The up/down helix motion direction is defined with the plane to reach. Up or down helix motion is defined with the difference between the current position and the plane to reach.

Sequential Groove Operation: Geometry (1/2)

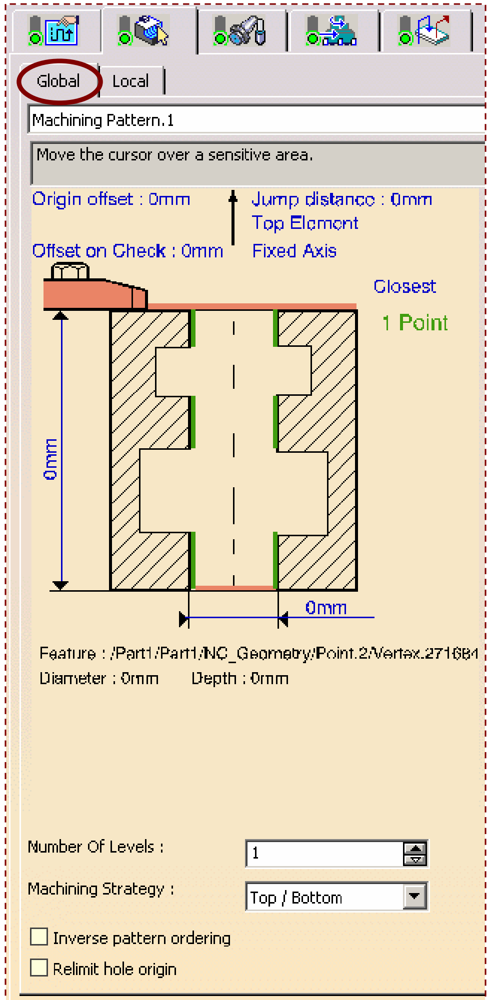
- Geometry selection is dedicated to groove machining.
- You can define planes, diameters (and offsets) or values to manage axial and circular motions.
- You can define up to 10 machining levels: 2 planes and 1 diameter is defined on each level.
- The sequential motions defined on the list are applied to each groove level.

Global:

The geometry (groove to machine) is managed in 'Global' tab.

- **Number Of Levels:** Allows definition of the number of grooves to machine (1Level = 1 Groove). You can define maximum 10 levels.
- **Machining Strategy:** Explains how the different levels are defined.
  - **Top/Bottom:** The first level to machine depends on the Plane 1 axial position. The first level to machine is the level on which Plane 1 is the upper one.
  - **Bottom/Top:** The first level to machine depends on the Plane 2 axial position. The first level to machine is the level on which Plane 2 is the lower one.

The level to machine is determined according to the distance from hole origin to the Plane 1 of a level.



Student Notes:

Student Notes:

Sequential Groove Operation: Geometry (2/2)

Local:

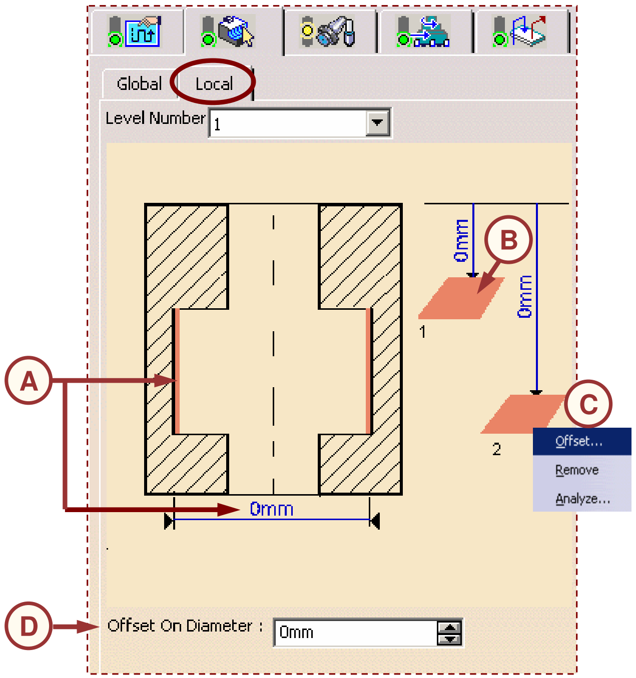
The additional geometry (planes or depth and offsets) is managed in 'Local' tab page.
For each machining level defined in the 'Global' tab page, geometry linked to that machining level is displayed.

A. Machining Diameter: You can specify a groove diameter value or select an element (circular edge).

B. You can select element or define depth diameter value

C. You can define Offset on the selected element (machining plane) by right-clicking on it.

D. Offset On Diameter: You can define the offset to be used on the defined groove diameter (or selected circular element).

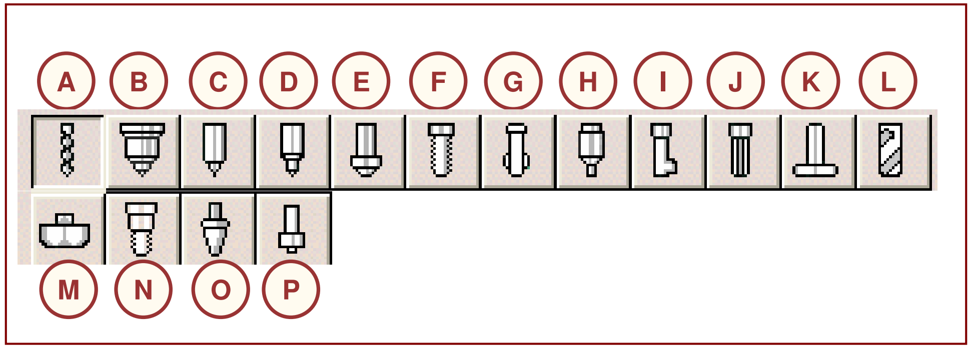


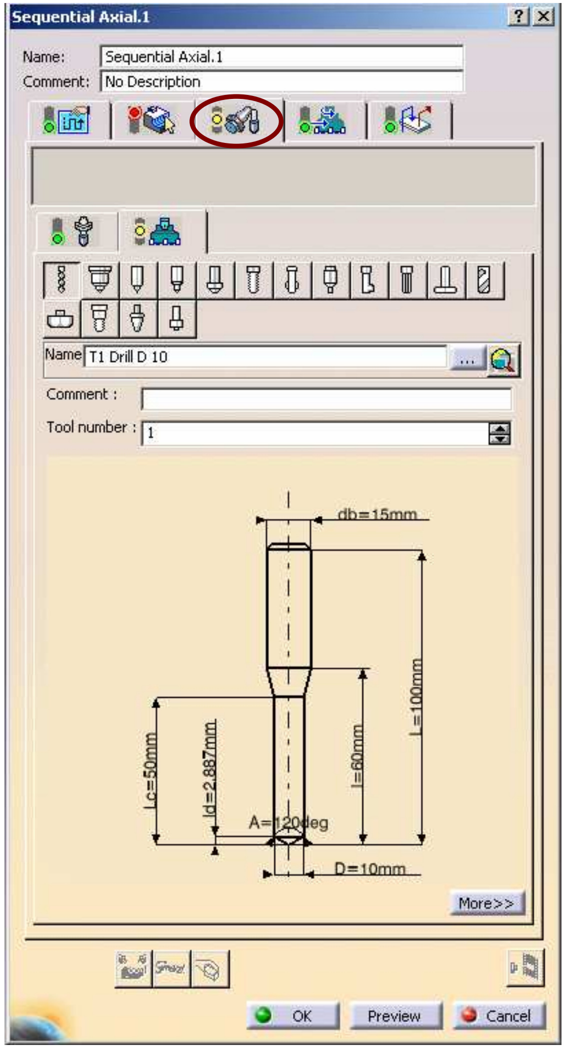
Positive value defines an offset inside the diameter and negative value defines an offset outside the diameter.

Student Notes:

Sequential Operations: Tools

You can use all Milling and Drilling tools for Sequential Axial Operation.





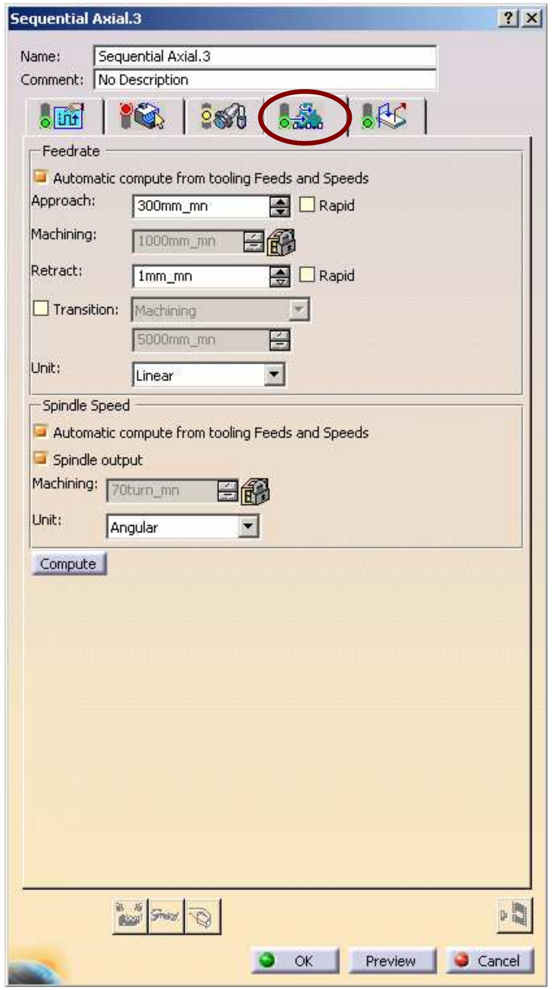
- **A. Drill**
- **B. Multi Diameter Drill**
- **C. Spot Drill**
- **D. Center Drill**
- **E. Countersink**
- **F. Tap**
- **G. Two Sides Chamfering**
- **H. Boring and Chamfering**
- **I. Boring Bar**
- **J. Reamer**
- **K. T- Slotter**
- **L. End Mill**
- **M. Face Mill**
- **N. Thread Mill**
- **O. Conical**
- **P. Counterbore**

STUDENT GUIDE

Student Notes:

Sequential Operations: Feeds and Speeds

You can define Feedrate and Spindle speed using this tab page. The Feedrate and Speed values can be computed automatically.



Machining, Approach, and Retract feedrates, and Spindle speed can be defined.

Spindle speed is applied on the different motions of the operations (including approach, retract, linking macros). Spindle can be re-defined with Spindle tool motion.

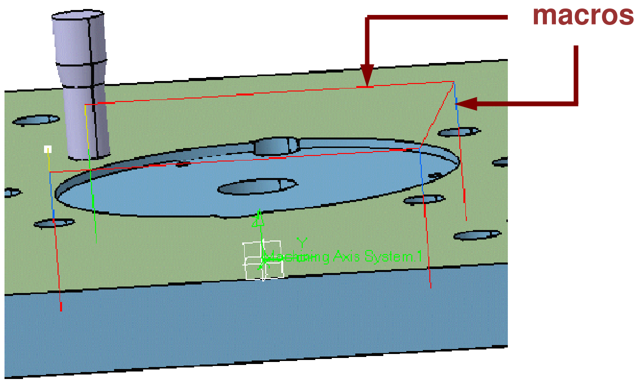
Cutting conditions (feed/tooth and cutting speed) can be included in a tools catalog. This data is converted into machining feedrate and spindle speed parameters to be used in machining operations by means of formula.

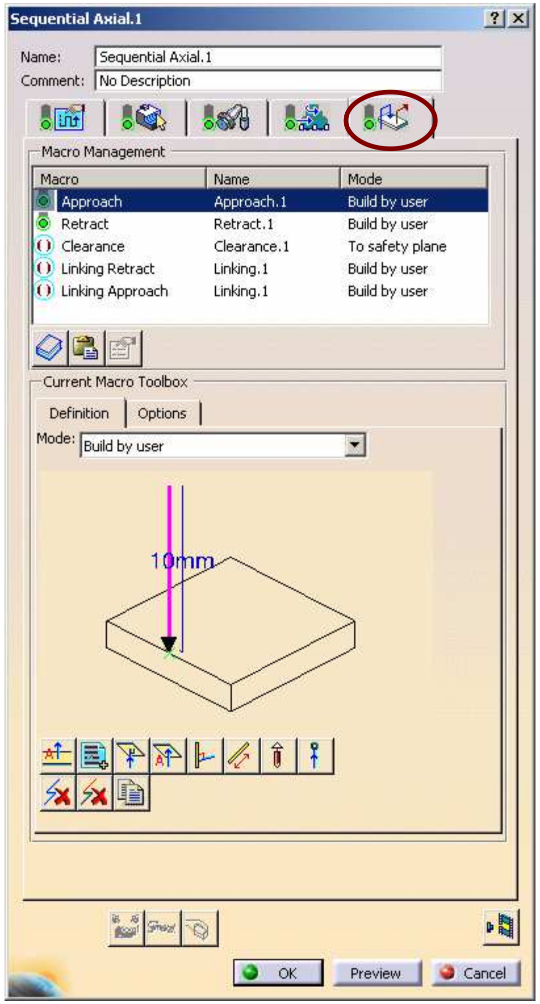
Sequential Operations: Macros

Following macros are available for Sequential Axial Operation:

- **Approach**
- **Retract**
- **Clearance, which can be used to define the feedrate on the horizontal path between two machining positions.**
- **Linking (between machining holes of the same pattern)**

All types of macros used in Drilling Operations are collision checked. If a check element is specified between two machined positions, a linking macro is applied to avoid collisions.





If a jump distance is defined on the operation, it will be used in preference to the linking macro.

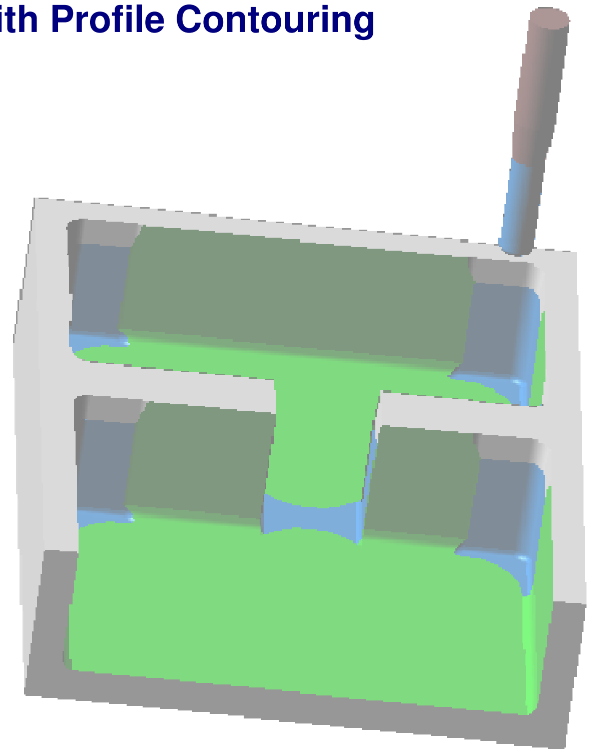
Student Notes:

Student Notes:

Prismatic Rework Area

In this lesson, you will learn what is Prismatic Rework area and how to use it for machining.

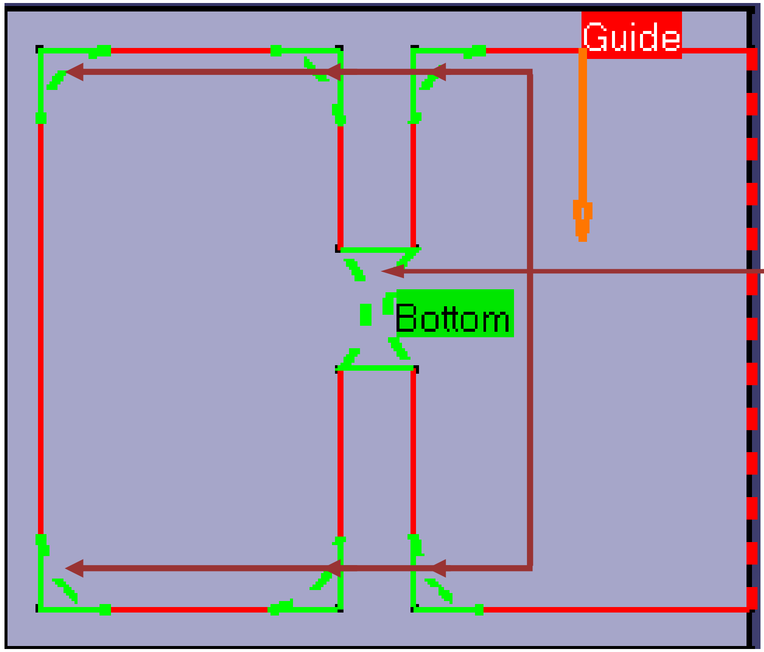
- Introduction
- Creating Prismatic Rework Area
- Prismatic Rework Area with Pocketing
- Prismatic Rework Area with Profile Contouring



Prismatic Rework Area: Introduction

Prismatic Rework Area is the area that is remained unmachined after the use of the previous operation tool. Using Prismatic Rework Area, you will know which area is required to be remachined using next suitable tool.

- **This functionality allows you to find area that is not machined from a previous operation or manual parameters.**
- **Thus you can use this area with a pocketing or a profile contouring operation.**



One channel and 6 corners shown in green in the shown picture are the Prismatic Rework Areas.

Student Notes:

Student Notes:

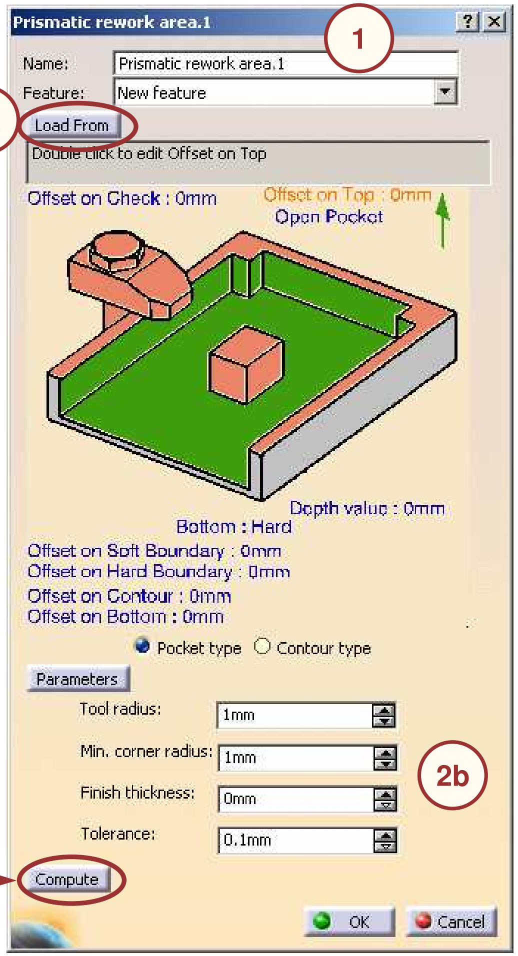
Creating Prismatic Rework Area: General Process

1 **Type the Name of the Operation. (Optional because a default name is given by the system 'Type_Of_Operation.X')**

2a **Select the operation which will be used as input to compute the area that is not machined**

or

2b **Type manually geometry and tool parameters then compute the area that is not machined**

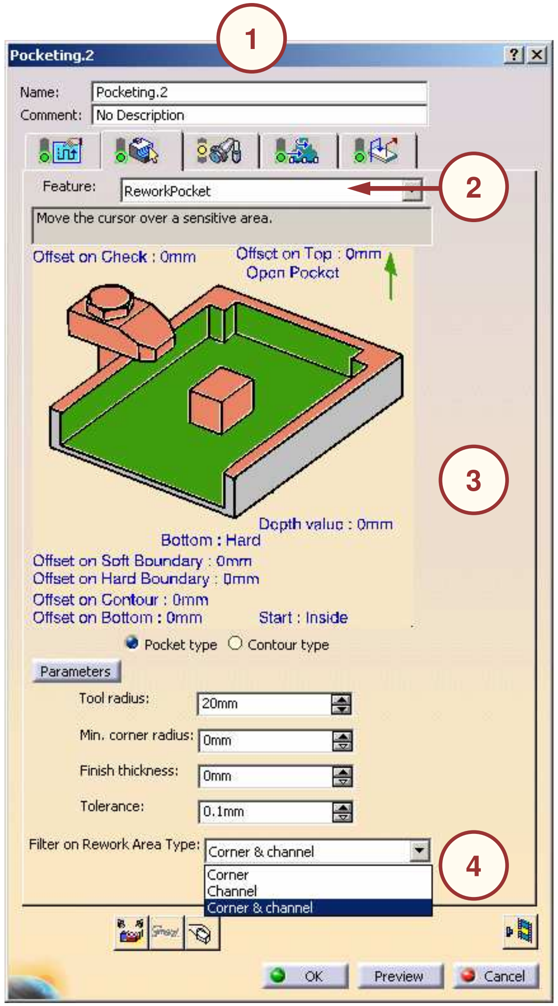


If you change your values, do not forget to compute it.

Student Notes:

Creating Prismatic Rework Area with Pocketing

1. Insert a Pocketing Operation
2. Select the area previously computed
3. All geometry and associated parameters are selected
4. You can choose corner and/or channel rework

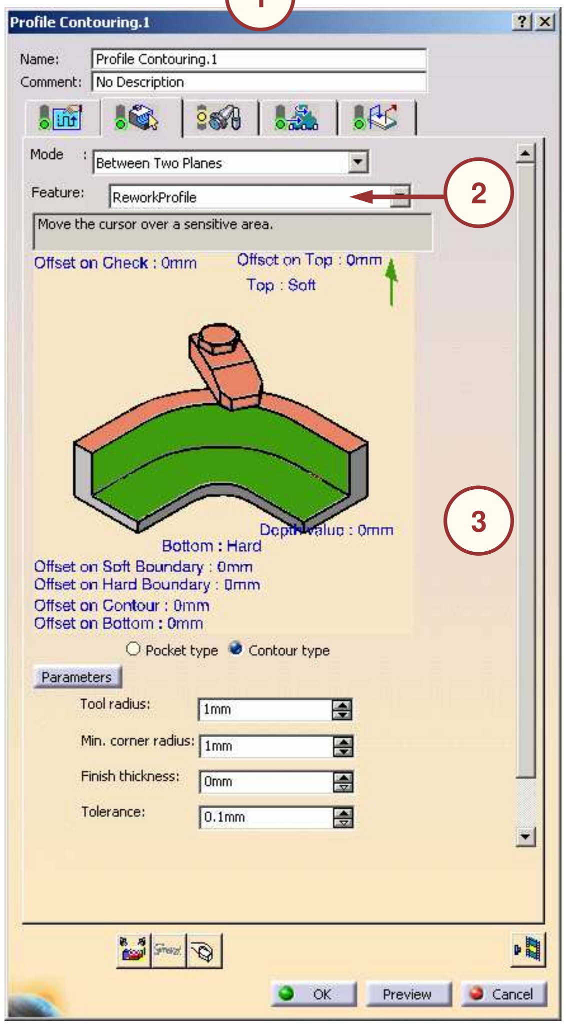


Rework area is also available in « Mfg View »
Sort by machinable features.

Student Notes:

Creating Prismatic Rework Area with Profile Contouring

1. **Insert a Profile Contouring Operation**
2. **Select the area previously computed**
3. **All geometry and associated parameters are selected**

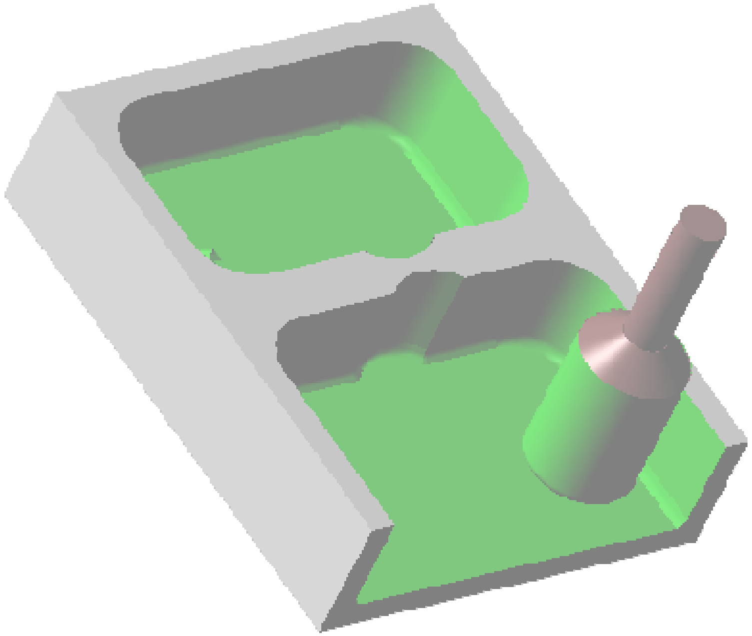


Only corner rework is available with Profile Contouring.

Prismatic Machining Area

In this lesson, you will learn how to use the created Prismatic Machining Area.

- Introduction
- Creating a Prismatic Machining Area with Pocket Type
- Using Prismatic Machining Areas for Pocketing Operation
- Creating a Prismatic Machining Area with Contour Type
- Using Prismatic Machining Areas for Profile Contouring Operation
- Using Prismatic Machining Areas for Machining Processes and Rework Areas



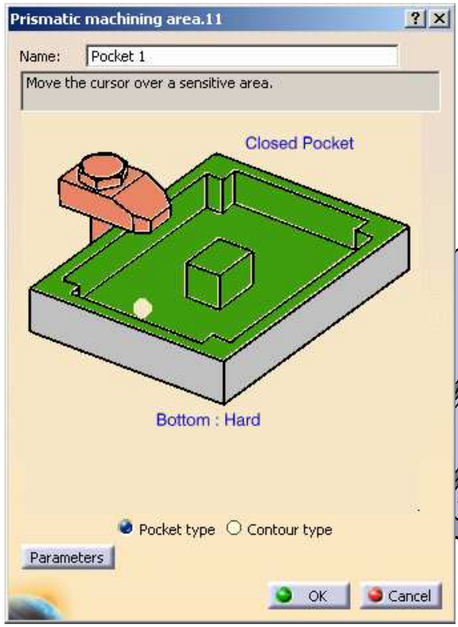
Student Notes:

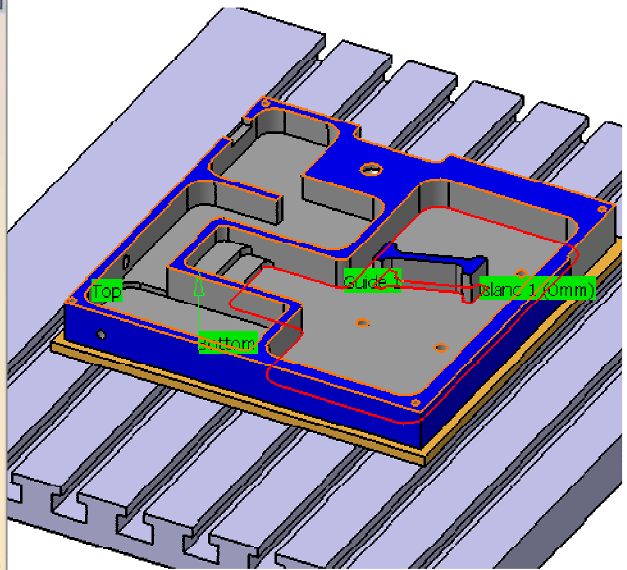
Student Notes:

Prismatic Machining Area: Introduction

Prismatic Machining Area is the area selected from Prismatic Rework for machining, using Pocketing or Profile Contouring Operation.

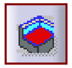
- **This functionality allows you to define an area from your geometry and record it.**
- **You can use this area with a pocketing or a profile contouring operation.**
- **Thus you can use this area with Machining processes and Prismatic Rework Area.**





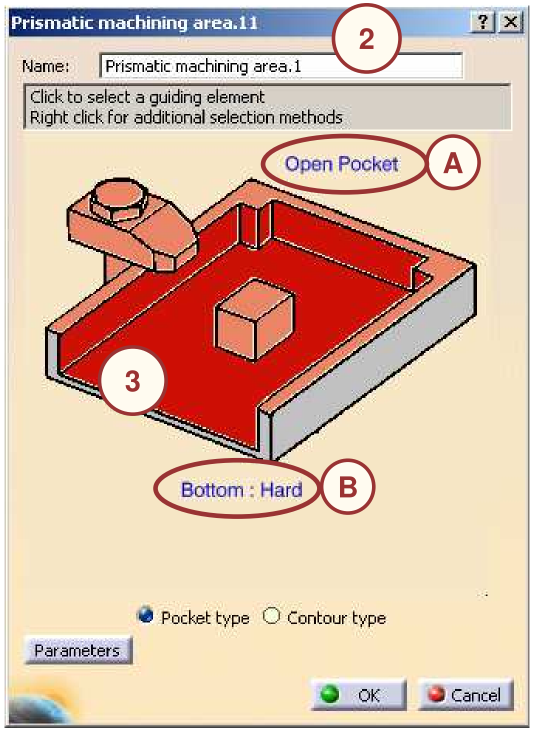
Student Notes:

Creating Prismatic Machining Area with Pocketing: General Process

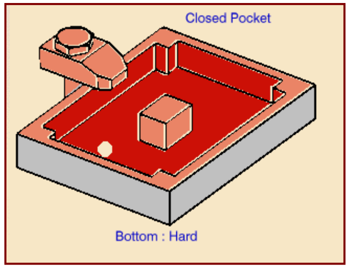
1. Click the Prismatic Machining Area icon 

2. Type the Name

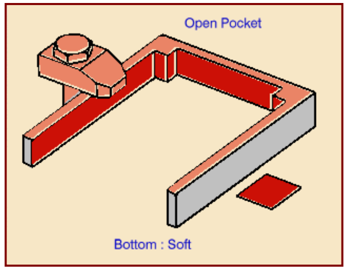
3. Click the red Bottom in the icon,then select the pocket bottom in the 3D window.
The pocket boundary is automatically deduced from the pocket bottom.



To deal with a closed pocket click: A



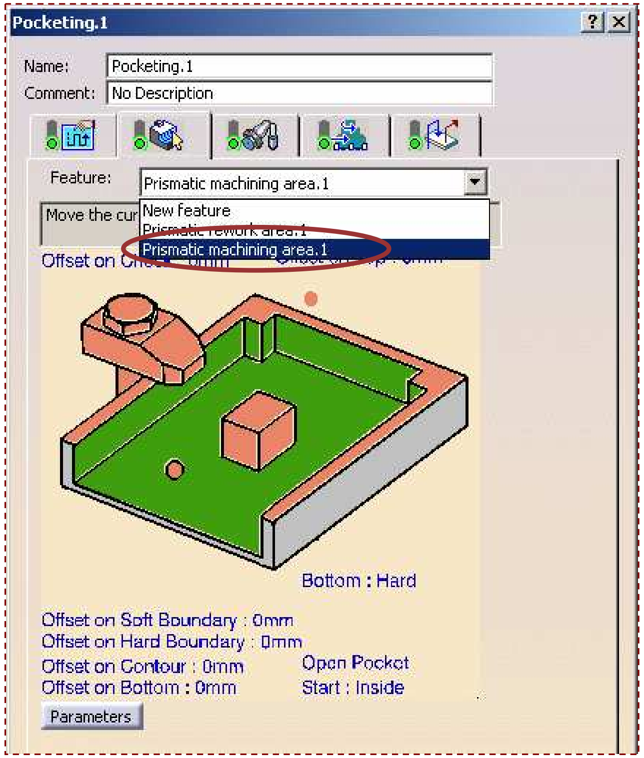
To remove the bottom (Soft Bottom) click: B



Student Notes:

Using Prismatic Machining Area for Pocketing Operation

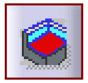
To perform a Pocketing Operation, click the Pocketing icon and instead of selecting a new feature, now you can use the already created Prismatic Machining area.



Prismatic machining area allows you to sort your design by Machinable Features thus you can spell it for instance in your Machining process.

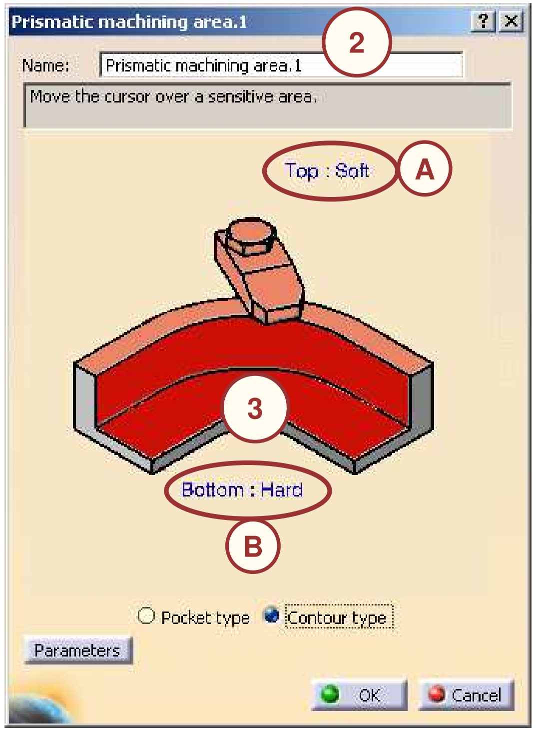
Student Notes:

Creating Prismatic Machining Area with Contouring: General Process

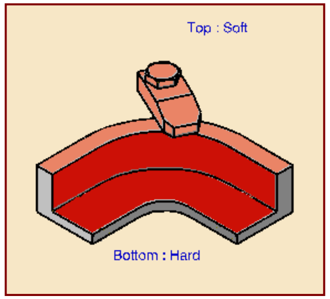
1 Click the Prismatic Machining Area icon 

2 Type the Name

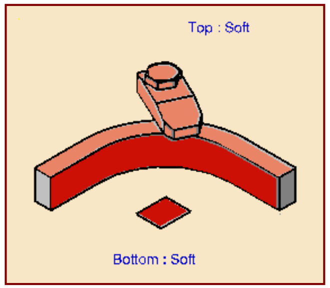
3 Click the red Bottom in the icon,then select the pocket bottom in the 3D window.
The pocket boundary is automatically deduced from the pocket bottom.



To add a Top (Hard Top) click: A



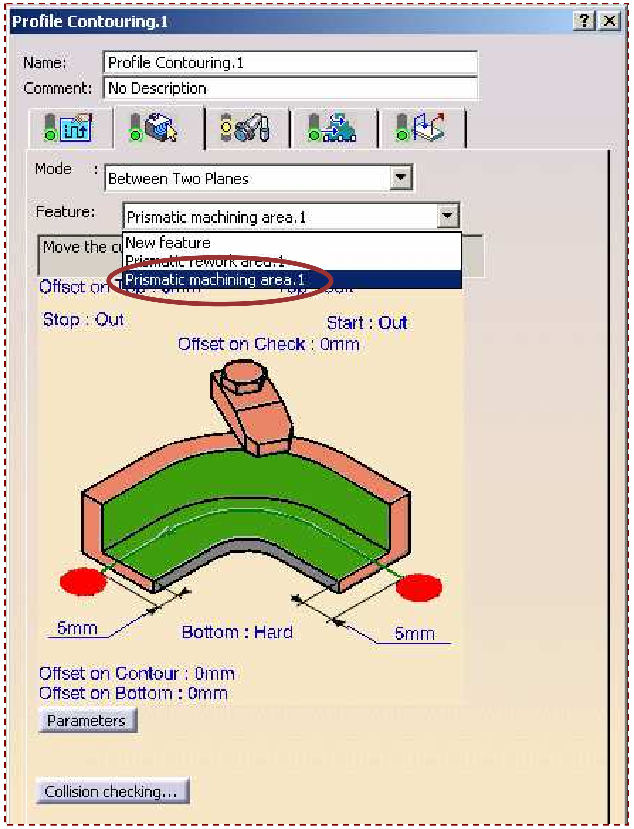
To remove the bottom (Soft Bottom) click: B



Student Notes:

Using Prismatic Machining Areas for a Profile Contouring Operation

To perform a Profile Contouring Operation, click the Profile Contouring icon and instead of selecting a new feature, now you can use the already created Prismatic Machining area.



STUDENT GUIDE

Student Notes:

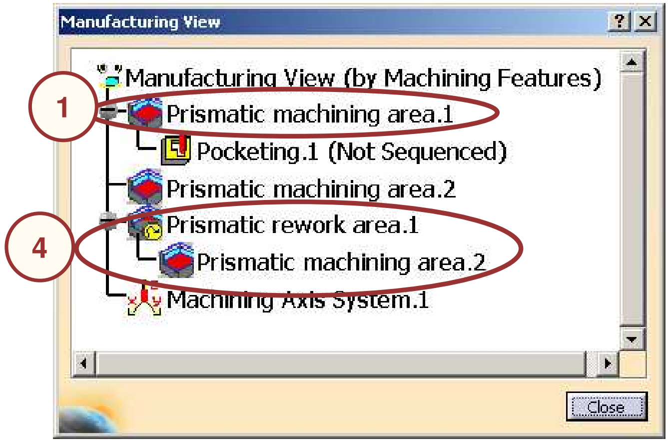
Using Prismatic Machining Areas for Machining Processes and Rework Areas

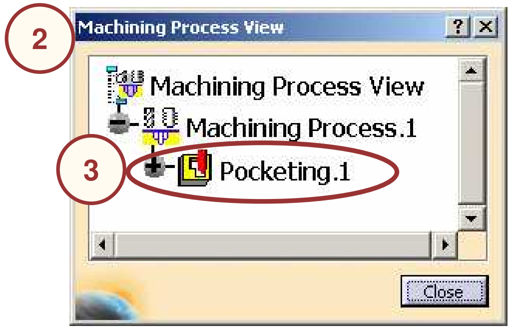
Prismatic Machining Areas can be used in Machining Processes (see the example below)

- **with Pocketing and Profile Contouring strategies**
- **including automatic Prismatic Rework Area generation**

1. **Create a Prismatic machining area**
2. **Open Machining Process View window**
3. **Create a machining process with a Pocketing operation for instance and select for geometry the Prismatic machining area**

   Check in Manufacturing View window
4. **Create a Prismatic Rework Area and select for geometry the Prismatic machining area**





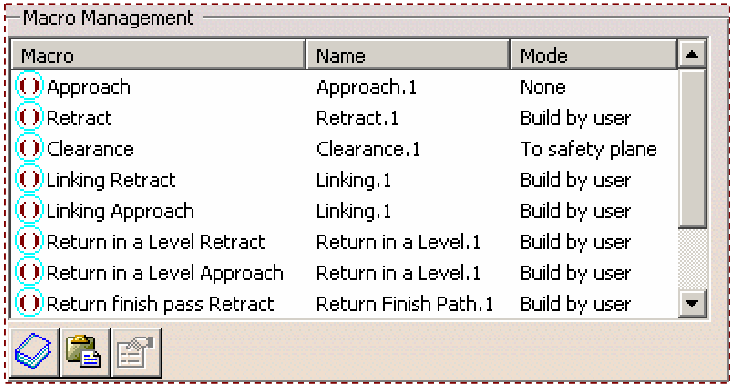
Prismatic Machining area allows you to manage features in your Machining processes; here the Pocketing operation uses both a Prismatic Machining Area and the Rework Area.

Student Notes:

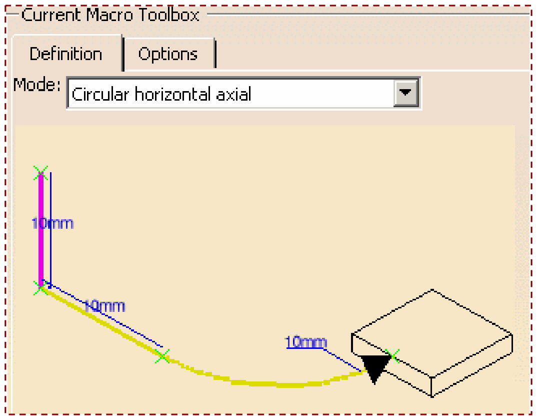
NC Macro Definition

In this lesson, you will learn how to create NC Macros.

- Introduction
- Types of Macro
- Details of Clearance Macro
- Pre-defined Macros
- Build by user Macros Tool Box
- Actions on Macros
- Create Macro with cutter compensation


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

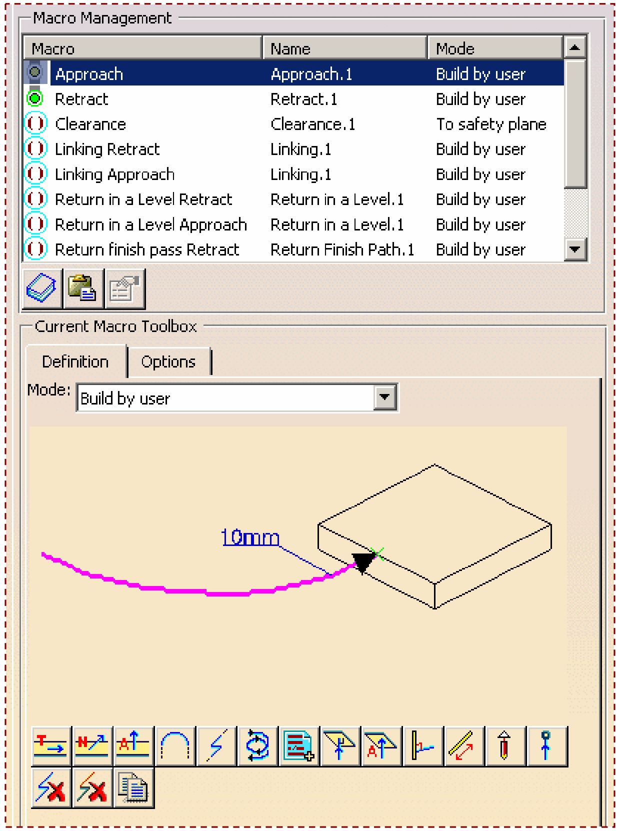


Student Notes:

NC Macro: Introduction

Macros are the tool motions outside the stock material that is required to be machined.

- **The NC Macro option provides features that enhance productivity.**
- **The non-working motions are controlled by macros.**
- **Tool damages either by collision or plunging are avoided using macros.**
- **Different types of macros are used according to the machining processes.**



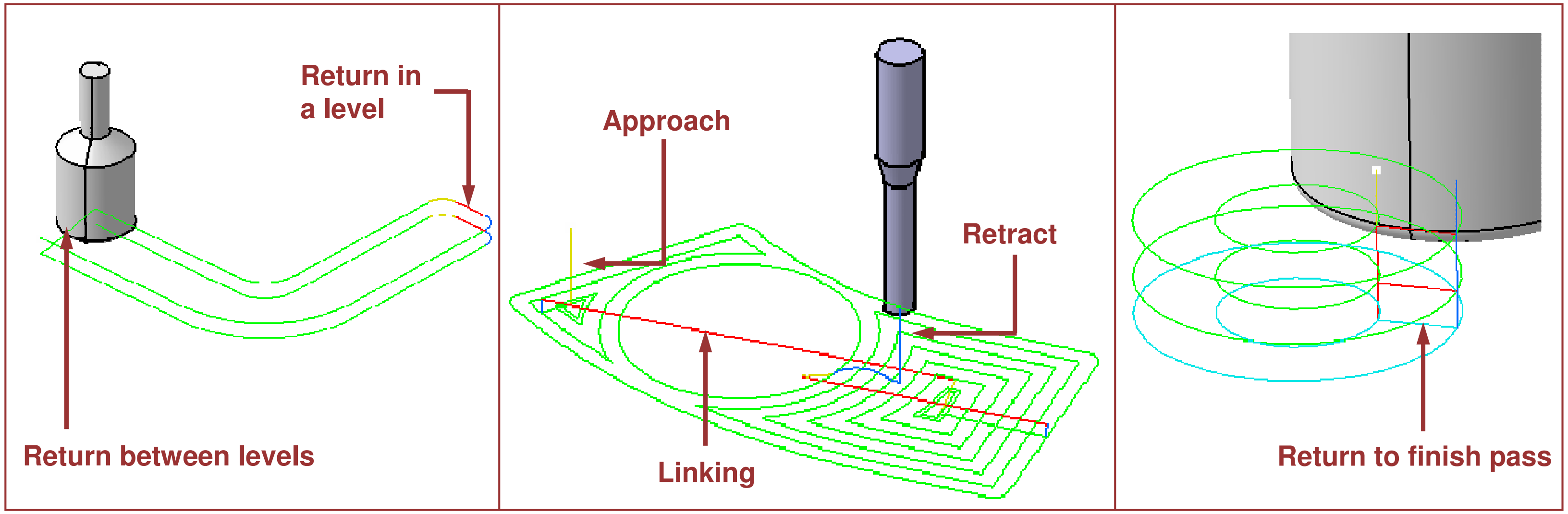
You can use the pre-defined macros or you can create your own macro as per the requirement.

Student Notes:

Types of Macro

There are seven different types of macros available as below:

- **Approach:** Before the first machining motion to enter into the material
- **Retract:** After the last machining motion to leave the part
- **Return between levels:** To move from the end of a level (retract) to the beginning of the next level (approach)
- **Return in a level:** In One-Way strategy allow to go from a pass to the next one (retract/approach) in the same plane
- **Linking:** To move inside a level from a sub-domain to another one (in case of collision, to avoid islands)
- **Return to finish pass:** To move to the finish pass with a retract motion then an approach motion
- **Clearance:** Special motion between each retract/approach motion of macros (see next page)



Student Notes:

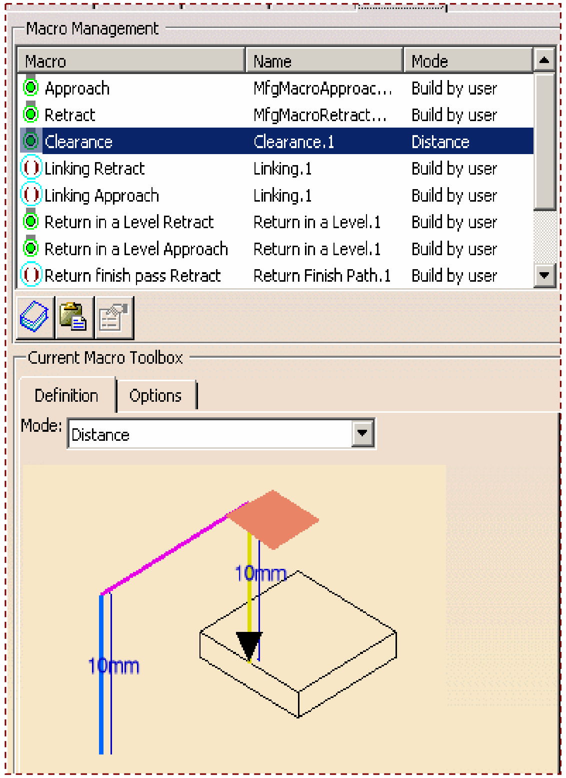
Details of a Clearance Macro

Clearance Macro:

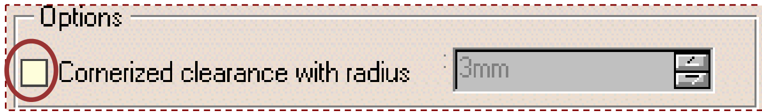
Each of the following macros

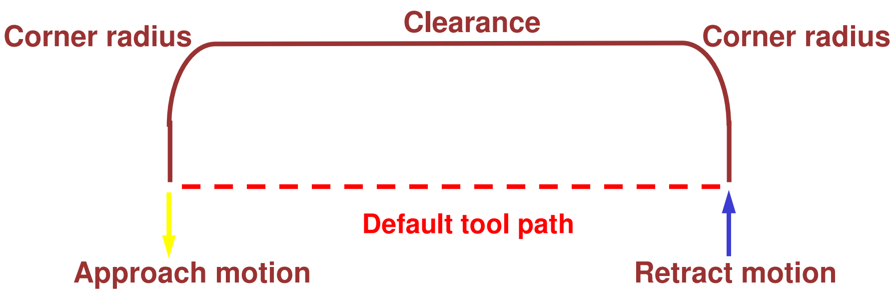
- **Return between level**
- **Return in a level**
- **Return to finish pass**
- **Linking**

is mainly divided in two motions: Approach and Retract.
Between these two motions, the system computes a default tool path.
If you want this transition tool path to be customized, then activate Clearance Macro.



You can Cornerize clearance with radius using option below:





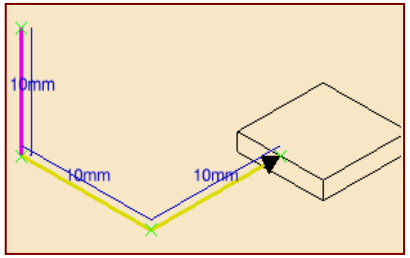
Student Notes:

Pre-defined Macros

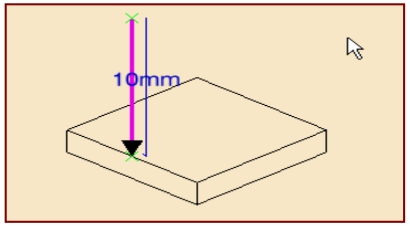
Depending on the type of macro you have selected, different types of Pre-defined Macros are available:

Example with Approach Macro:

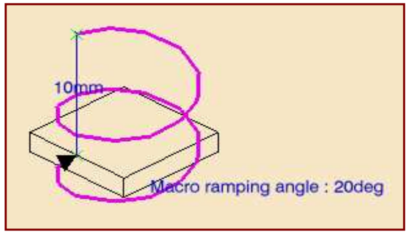
Horizontal horizontal Axial

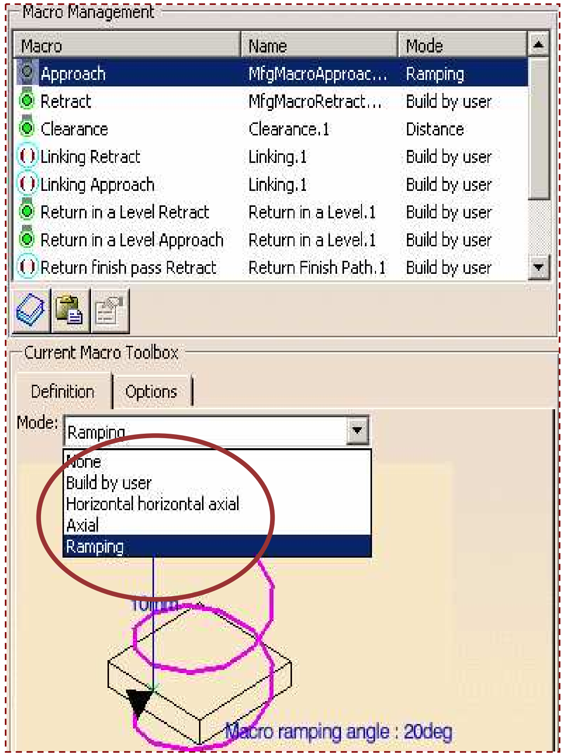


Axial



Ramping approach



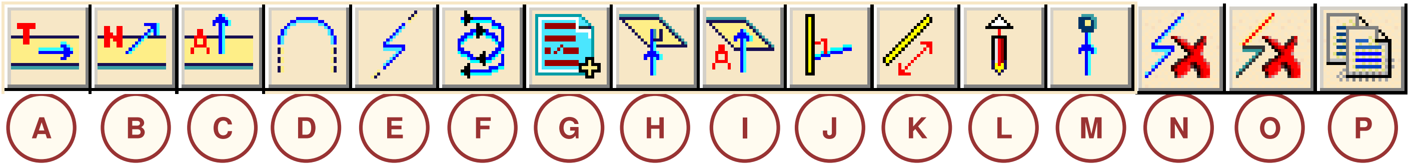


You can define your own macro with Build by user menu.

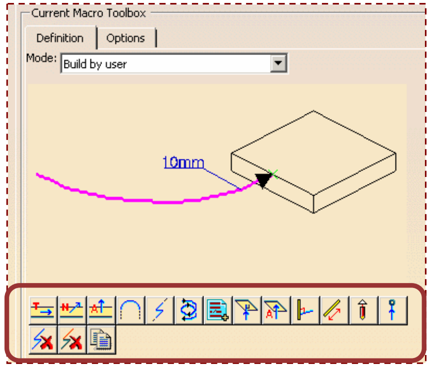
Student Notes:

Build by user Macros Tool Box

Tool box to create your own Macro:



- **A** Tangent
- **B** Normal
- **C** Axial
- **D** Circular
- **E** Ramping *
- **F** Helix Approach Motion
- **G** Insert PP word
- **H** Motion perpendicular to a plane
- **I** Axial motion up to a plane
- **J** Motion to a line
- **K** Distance along a line
- **L** Tool axis motion
- **M** Motion to a point
- **N** Remove all motions
- **O** Delete selected motions
- **P** Copy Approach or Retract macro on all approach or retract motions of the other macros

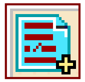


You can create different macros combining these basic paths in any order.

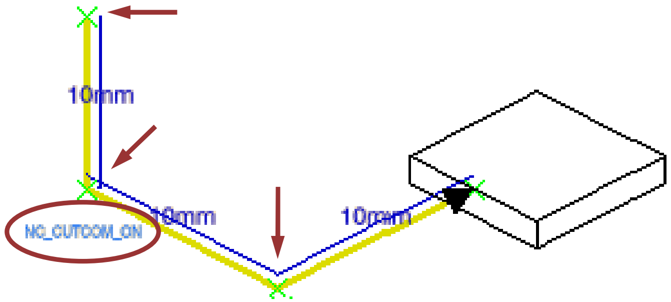
Ramping Macro has a dedicated option in pocketing operation.

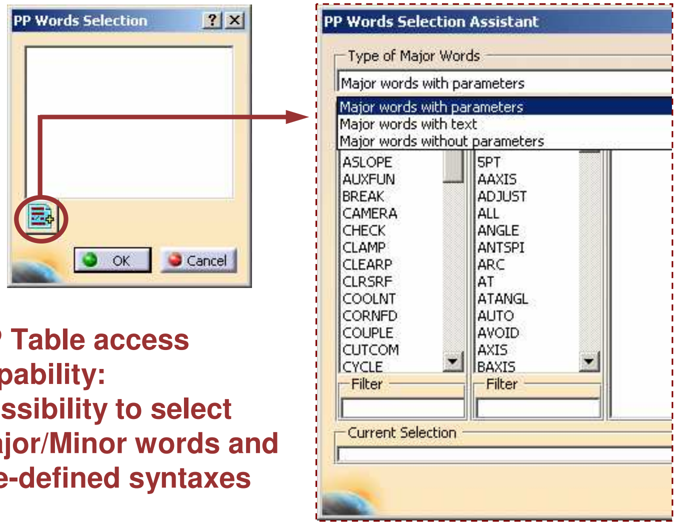
Student Notes:

Actions on your Macro

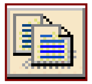


Insert a PP word on a point of the macro.
Cross symbols localize the possible points to insert the PP word
To insert a PP word, you can also press right mouse button on the cross and select « PP word list »

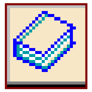




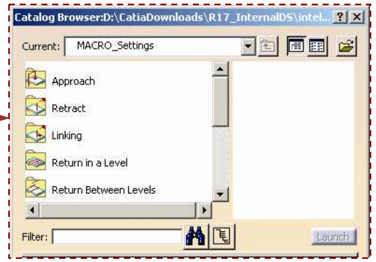
PP Table access capability: Possibility to select Major/Minor words and pre-defined syntaxes



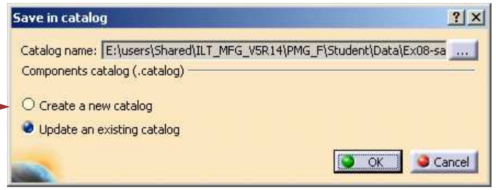
Apply this Approach or Retract motion to all Return and Linking Macros in the operation (only available on Approach macro and Retract macro)



You can read macro from a catalog



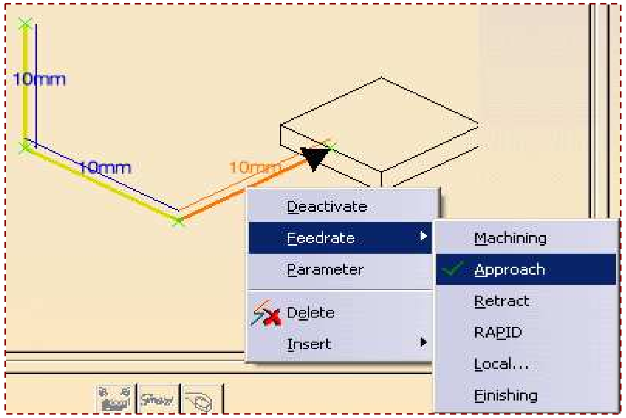
You can store macro in a catalog



Student Notes:

How to modify Parameters on Macro

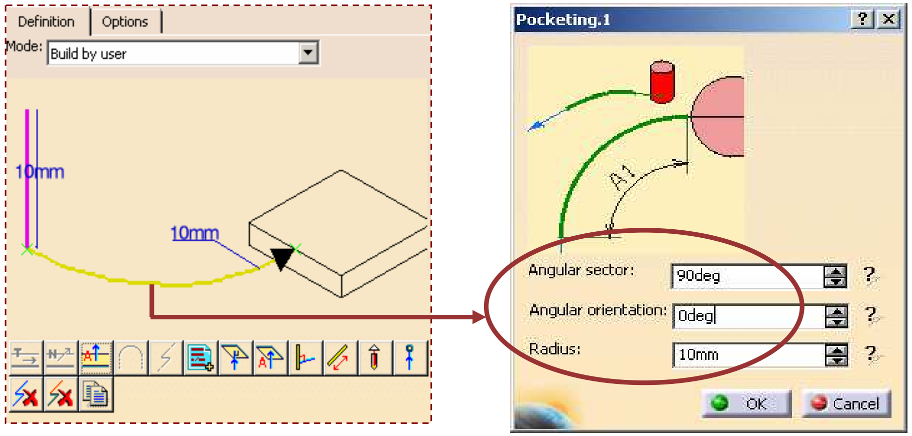
To locally modify a feedrate in the macro, right-click a element and select « feedrate » to choose which feedrate to associate between Machining, Approach, Retract, Rapid, Local or Finishing



Depending on the feedrate selected, the element takes a different color:

Yellow : Approach
White : Local
Green : Machining
Blue : Retract
Red : Rapid

To modify geometrical parameters of a macro, double-click it.



Student Notes:

Creating Your Own Macro with Cutter Compensation(1/2)

Cutter Compensation (CUTCOM instruction) can be generated for all operations which machine with the side of the tool.

The compensation can be defined:

- **On Approach, Retract, Return to finish pass, Return between levels macros.**
- **CUTCOM/LEFT or CUTCOM/RIGHT can be activated.**
- **CUTCOM/OFF is generated to deactivate the cutter compensation.**

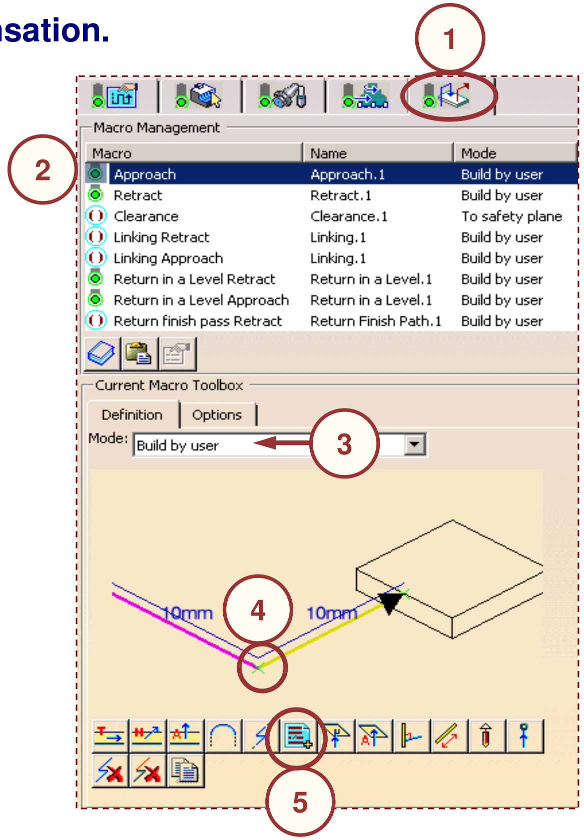
1 Select the Macro tab page in the operation

2 Select the macro type (Approach, Retract)

3 Select a mode between a predefined macro or a macro Build by user

4 Select the point where Cutter compensation must be inserted in the macro path

5 Click icon 'Add PP word list' or use 'PPword list' with the contextual menu on this point



Student Notes:

Creating Your Own Macro with Cutter Compensation(2/2)

6 Click icon 'To access PP table' in the PP Words selection dialog box (Note: this icon is accessible only if a machine and an associated PPtable are defined on the part operation)

7 Select the Major Word 'CUTCOM' in the displayed list (Major Words with parameters

8 Select the appropriate NC_CUTCOM syntax in the displayed list

NC_CUTCOM_ON → activates compensation. CUTCOM/LEFT or CUTCOM/RIGHT (The valuation LEFT/RIGHT is automatically defined by the system in order to respect the machining side)

NC_CUTCOM_OFF → deactivates Compensation. CUTCOM/OFF is generated

9 Click Apply. The syntax will appear in PP Words Selection and on 3D viewer.