Student Notes:

CATIA V5 Training
Foils

Advanced Part Machining

Version 5 Release 19
January 2009

EDU_CAT_EN_AMG_FF_V5R19

STUDENT GUIDE

Student Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to:

- **Identify and use the Advance Part Machining workbench tools**
- **Define a Multi-Axis Flank Contouring operation**
- **Define a Multi-Axis Helix Machining operation**
- **Define a Cavities Roughing operation**

Targeted audience

Advanced NC Programmers

Prerequisites

Students attending this course should have knowledge of Numerical Control Infrastructure (NCI), PMG, SMG and MMG workbench.

12 hours

Student Notes:

Table of Contents (1/2)

Student Notes:

Table of Contents (2/2)

Student Notes:

How to Use This Course

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

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

Recap Exercises:
Recap Exercises are provided along at the end of each lesson to reinforce the concepts learnt.

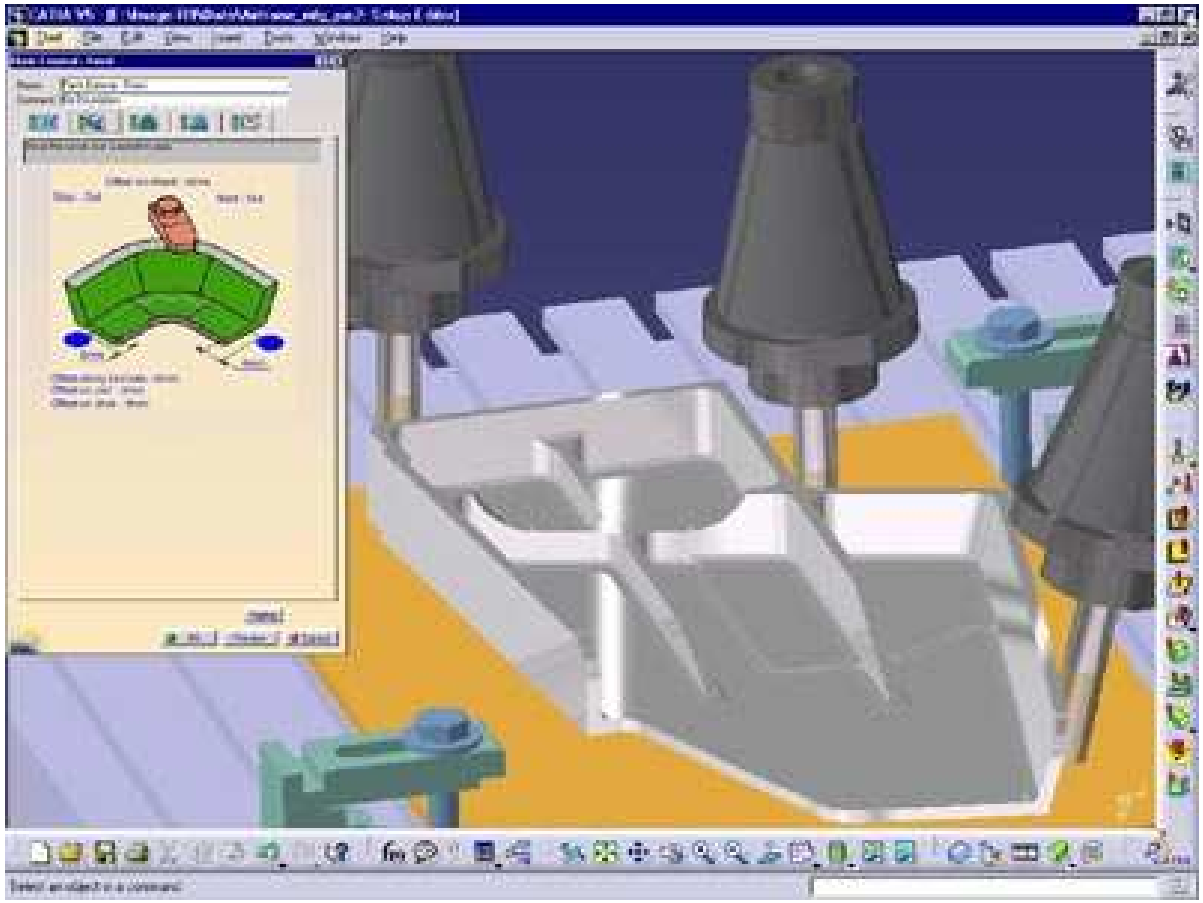
A Master Exercise:
A Master Exercise provides a project where an industry type part is used to assist you in applying the key knowledge and skills acquired in the individual lessons as they apply to real world scenarios.

Note*: T*he Master Exercise is provided at the end of the course to practice on key concepts in the lessons.

Student Notes:

Introduction to Advanced Part Machining

You will become familiar with the Advanced Part Machining.

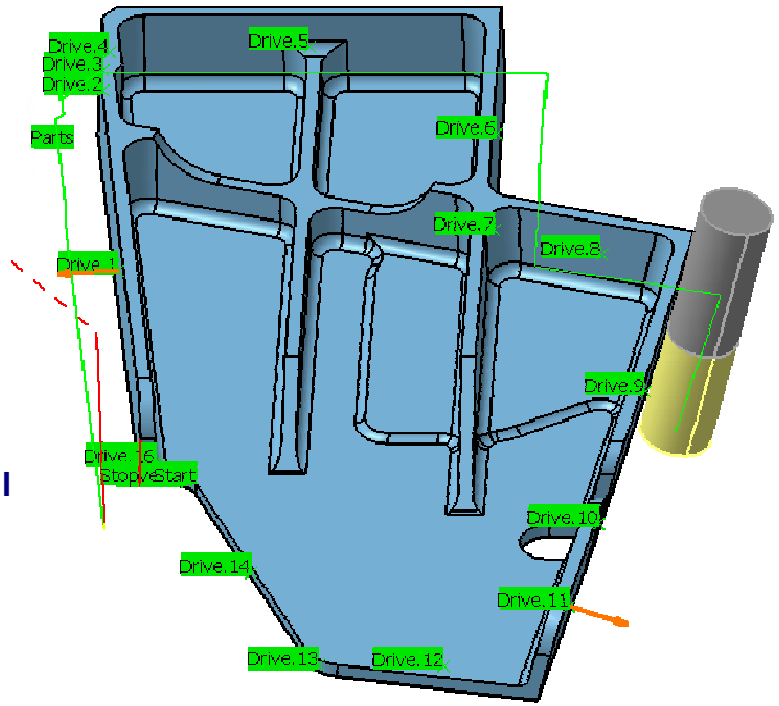


About Advanced Part Machining

Advanced Machining (AMG) workbench easily generates high quality NC programs for machining complex 3D parts and free form shapes. AMG is beneficial to machine aerospace, turbo-machinery, hydraulic and much more complex 3D parts, all in a single machining solution.

AMG includes 2.5 to 5-axis machining technologies and Axial machining. It brings new functionalities in order to cover the entire machining process, in addition to existing key functionalities in other machining solutions.

AMG develops machining strategies that optimize toolpaths, eliminate unnecessary air cutting, maximize tool life, reduce programming time and increase overall productivity.



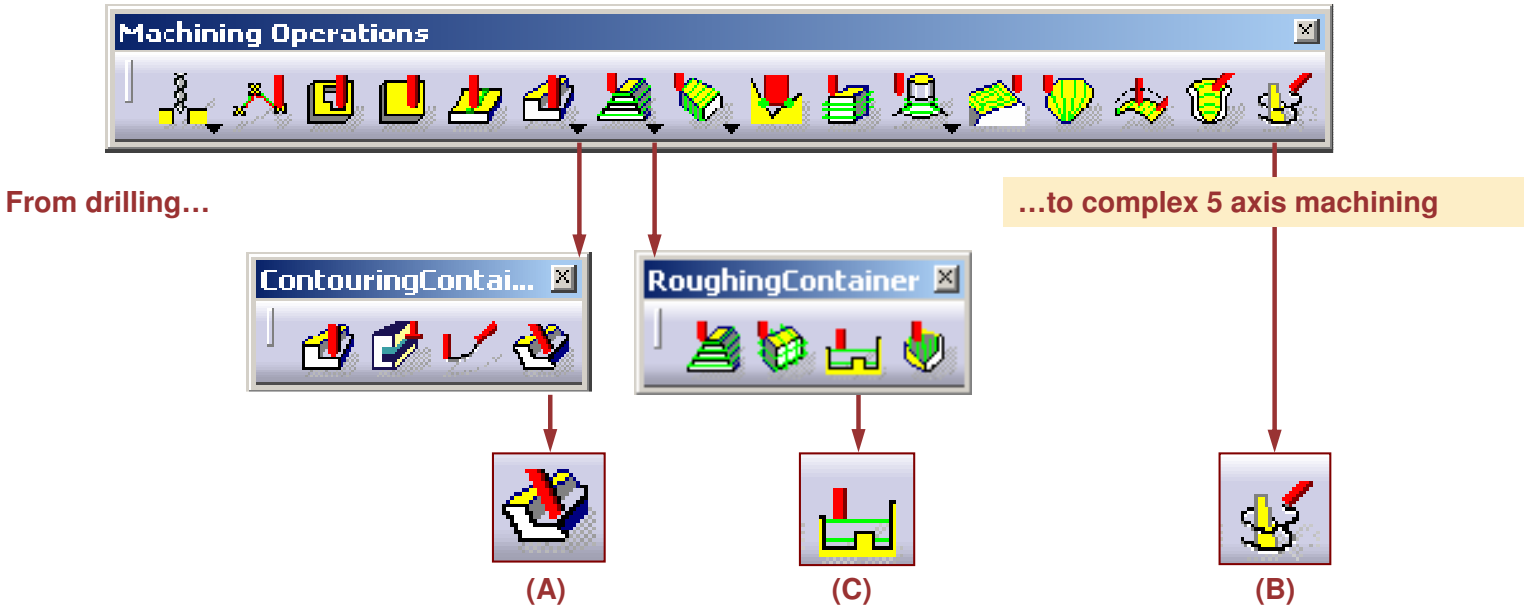
AMG benefits the user for:

- **Quick tool path generation**
- **Flexible management of tools and tool catalogs**
- **Definition of Machining areas**
- **Automated reworking**
- **Fast tool path update after modification**
- **Tool holder collision checking**
- **Quick verification of tool path**
- **Seamless NC data generation**

Student Notes:

Student Notes:

Operations in Advanced Machining Workbench



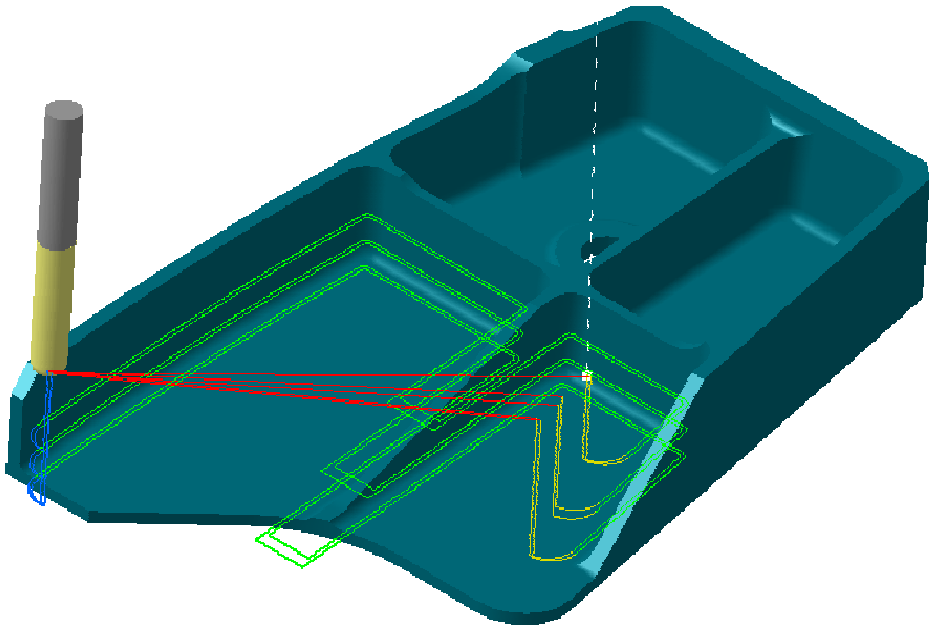
This training is focused on (A) Multi-Axis Flank Contouring, (B) Multi-Axis Helix Machining and (C) Cavities Roughing.

Refer to the PMG, SMG and MMG courses for learning the other operations of AMG workbench.

Multi-Axis Flank Contouring

You will become familiar with the principles of 5-Axis Flank Machining.

- About Multi-Axis Flank Contouring
- General Process
- Geometry
- Strategy
- To sum Up



Student Notes:

Student Notes:

About Multi-Axis Flank Contouring

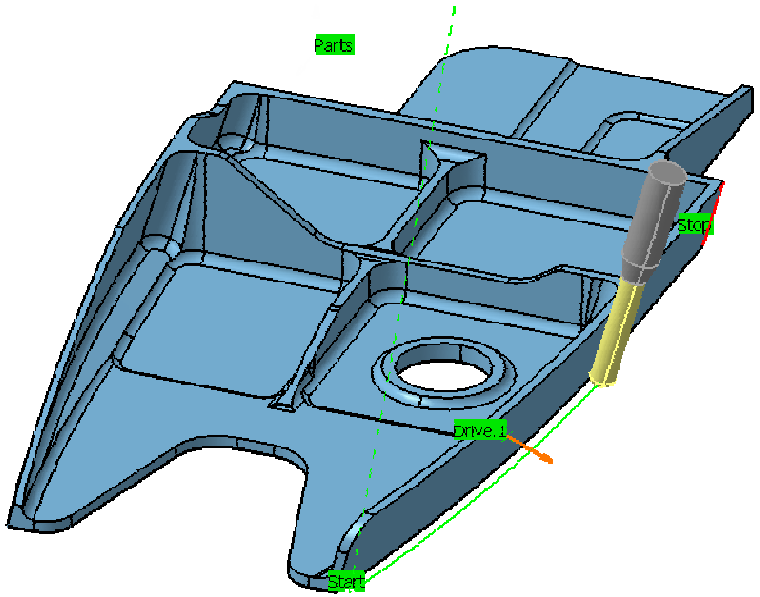
The Multi-Axis Flank Contouring operation is mainly used for semi-finishing and finishing of 5 axis walls in structural parts. In this operation the cutting tool machines with flank.

This is a profile contouring operation in which the tool axis can be changed according to the side to be machined, by using various strategies.

The Multi-Axis Flank Contouring operation is especially useful for machining the flanks of the structured parts used in the aerospace industry.

Collision with the drive surface can be avoided.

In Multi-Axis Contouring operation, you can change the local feedrates while machining along the area.



Student Notes:

Multi-Axis Flank Contouring: General Process

1. Specify the Name of the operation. (optional, because a default name is given by the system 'Type_Of_Operation.X')

2. Type text of comment, if needed.

3. Define the operation parameters using the 5 tabs.

Strategy tab

Geometry tab

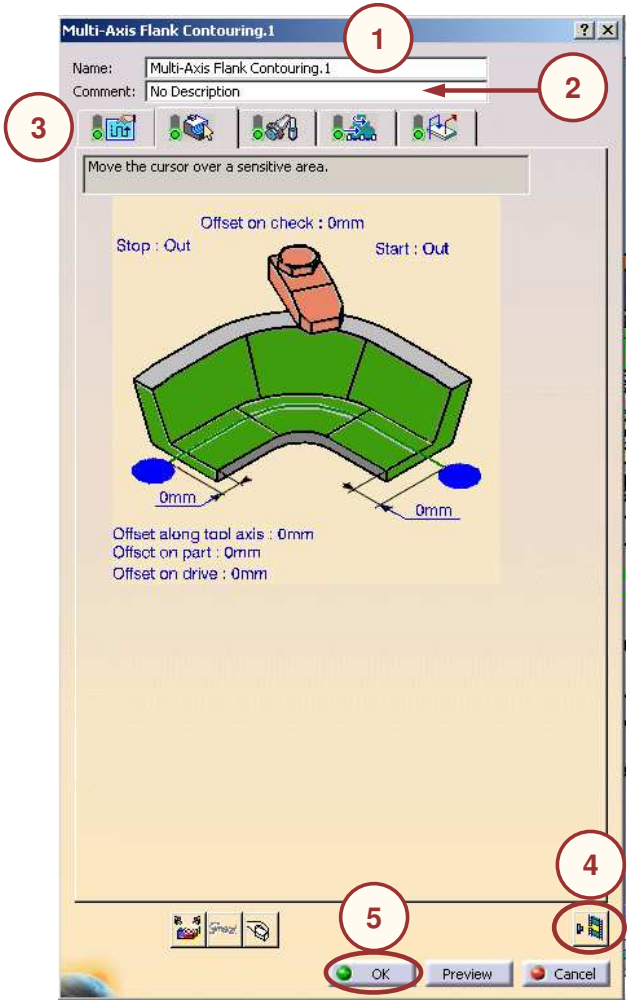
Tool Definition tab

Feeds & Speeds tab

Transition Paths tab

4. Replay and/or Simulate the tool path.

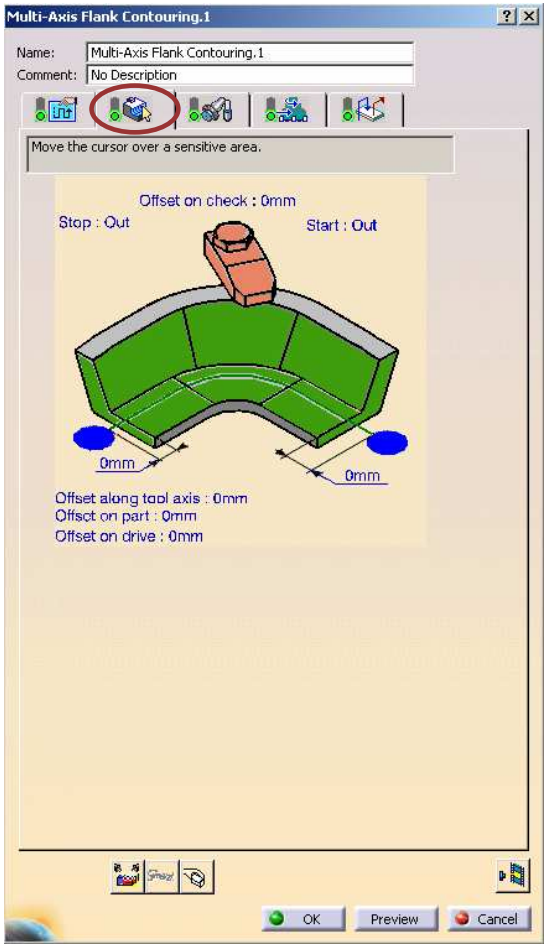
5. Click OK to create the operation.



Student Notes:

Multi-Axis Flank Contouring: Geometry

You will see the options on the Geometry tab of Multi-Axis Flank Contouring.

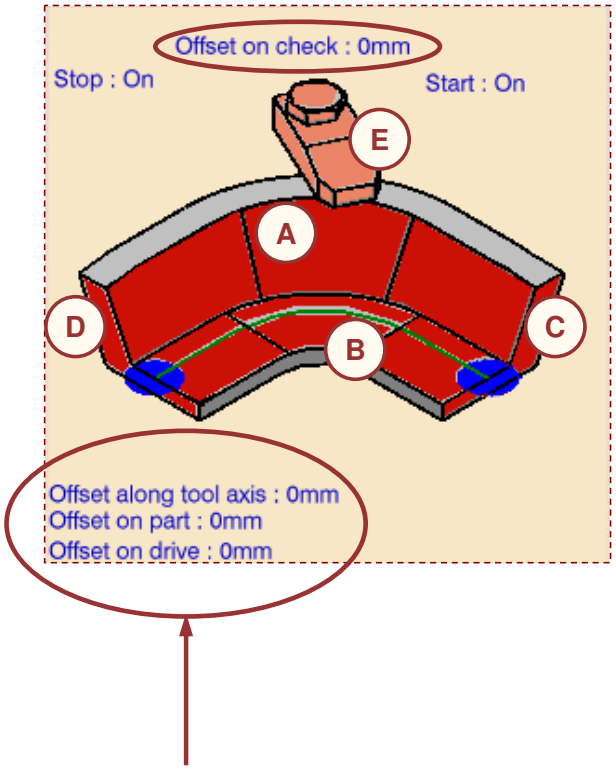


Student Notes:

Presentation

The Geometry tab includes a sensitive icon dialog box that allows the selection of:

- **Drive Surface elements** (A)
  - **Flank tool will lean on Drives**
    - **With respect to tool axis strategy and offset**
- **Part Surface elements** (B)
  - **Tool end will lay down on Part**
    - **With respect to tool axis strategy and offset**
- **Start element** (C)
  - **Used to compute the initial tool position**
- **Stop element** (D)
  - **Used to compute the final tool position**
- **Check elements (optional)** (E)
  - **Elements to avoid during machining**



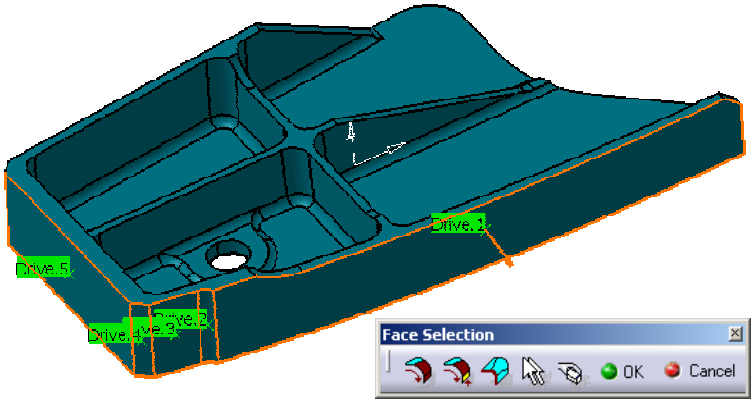
Offset can be applied on part, drive, check and along tool axis

Student Notes:

Drives Elements (1/3)

- **Face selection:**
  This wizard allows you to quickly select drives.

  - **To start the navigation, you always need to select at least two faces (first one is start element, second one gives the direction to navigate).**



  - **Then you can select Navigates on belt of faces: Navigation is done in order to follow a belt or**

  - **You can select Navigates on Faces Until a Face: Navigation is done until a selected face.**

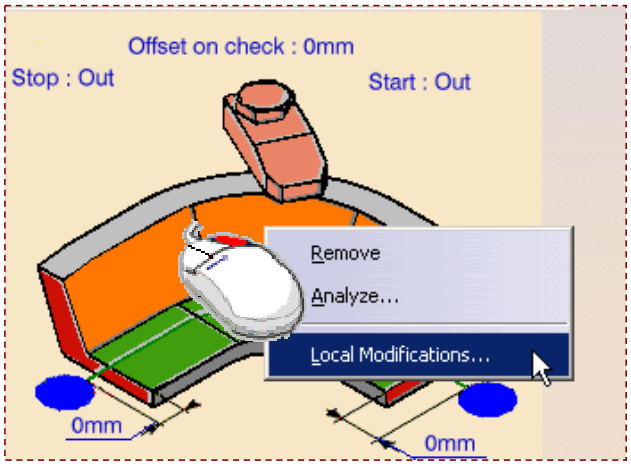
  - **You can use Multi-selection of Face: Multi-selection of faces is possible when selecting drive elements.**

Student Notes:

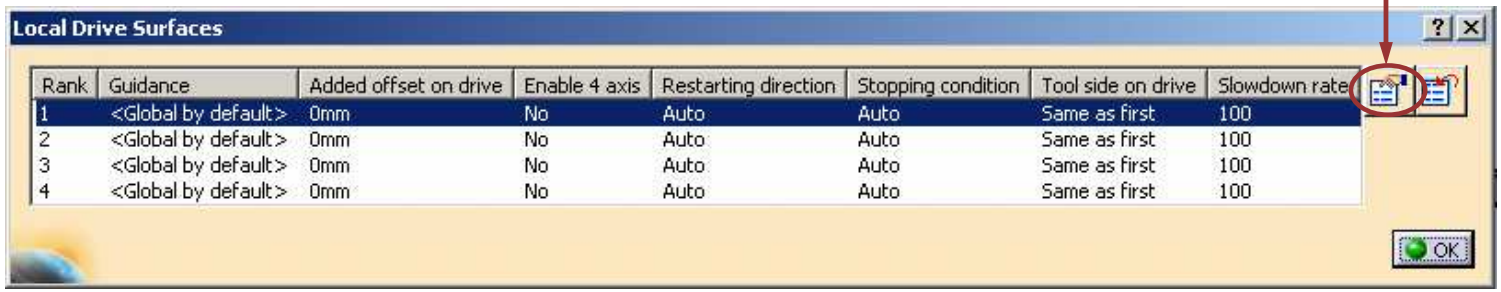
Drives Elements (2/3)

Local Modifications:
(contextual menu on drive elements)
This task illustrates how to locally modify a Multi-Axis Flank Contouring operation in the program.

- **Once all drives are selected, you can modify locally strategy and offset on each drives.**
- **Drive surfaces properties dialog box:**



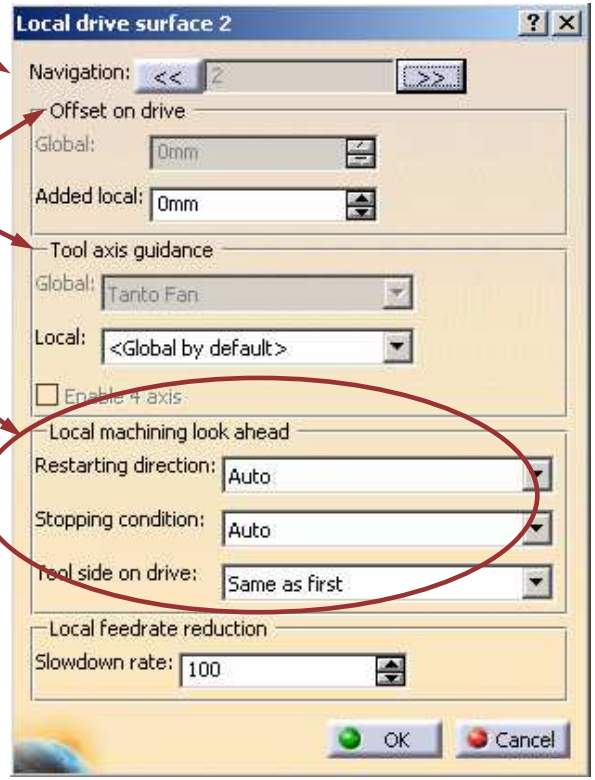
Select this icon to open the drive editor

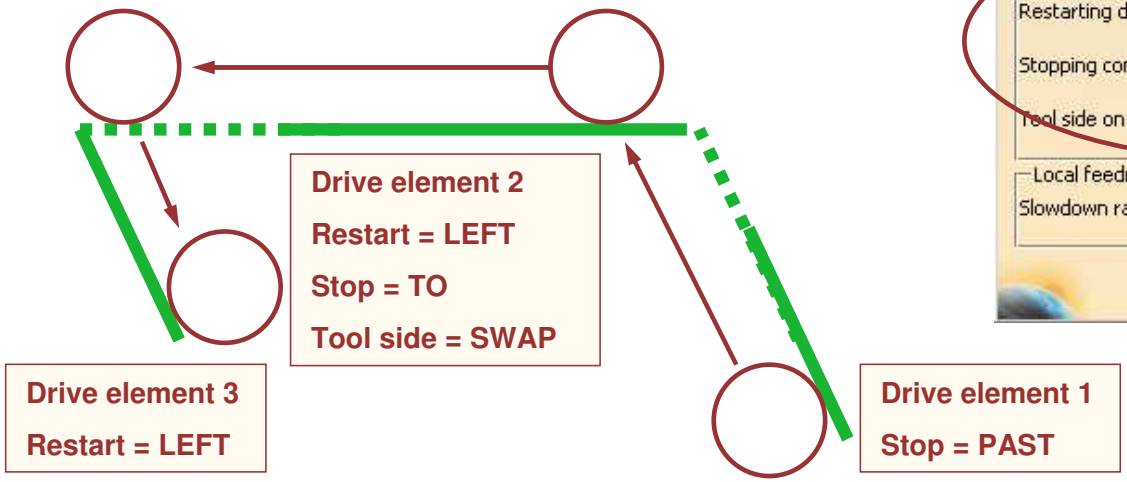


Student Notes:

Drives Elements (3/3)

- Local Modification Wizard
  - Browser on drive: select the drive on which you want to perform modification (selected drive is highlighted)
  - Local offset modification
  - Local Tool axis guidance modification
  - In case of non- contiguous drives ability to define locally on each drives Start, Stop and positioning conditions.

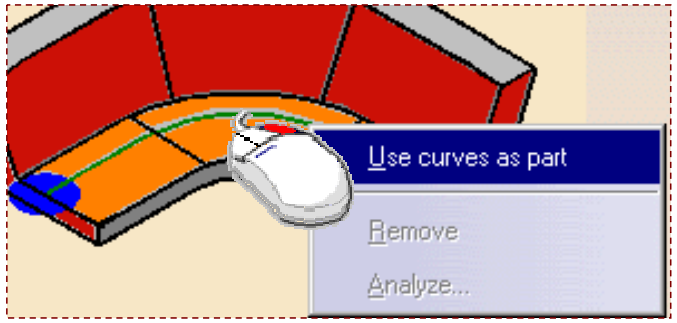




Student Notes:

Part Elements (1/2)

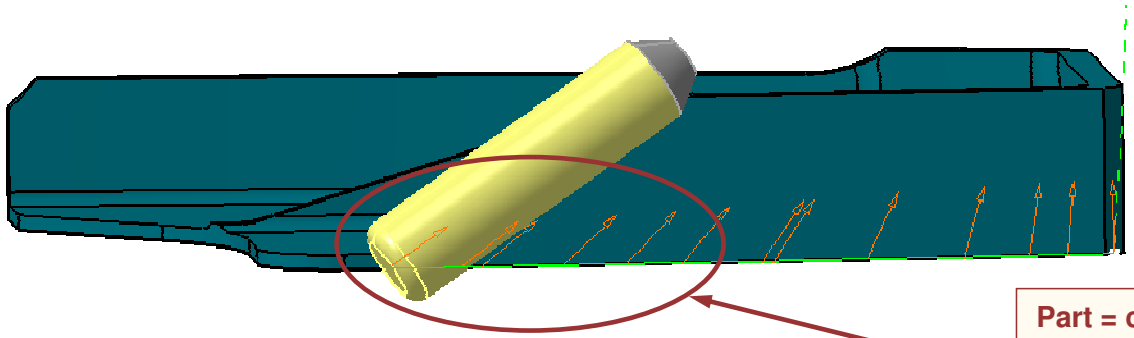
- Use curves as part:
  A curve can be selected as a Part surface.
  - Click right (MB3) on part element sensitive icon and select 'Use curves as part'
    - The system accepts only curves that are boundary of selected drives.
  - Edge selection:
    This wizard allows you to quickly select the curves
    - To start the navigation, you always need to select at least two edges (first one is start element, second one give the direction to navigate).
    - Then you can select
      Navigates on belt of edges:
      Navigation is done in order to follow a belt or 
    - You can select
      Navigates on Edges Until an Edge:
      Navigation is done until a selected face 



Student Notes:

Part Elements (2/2)

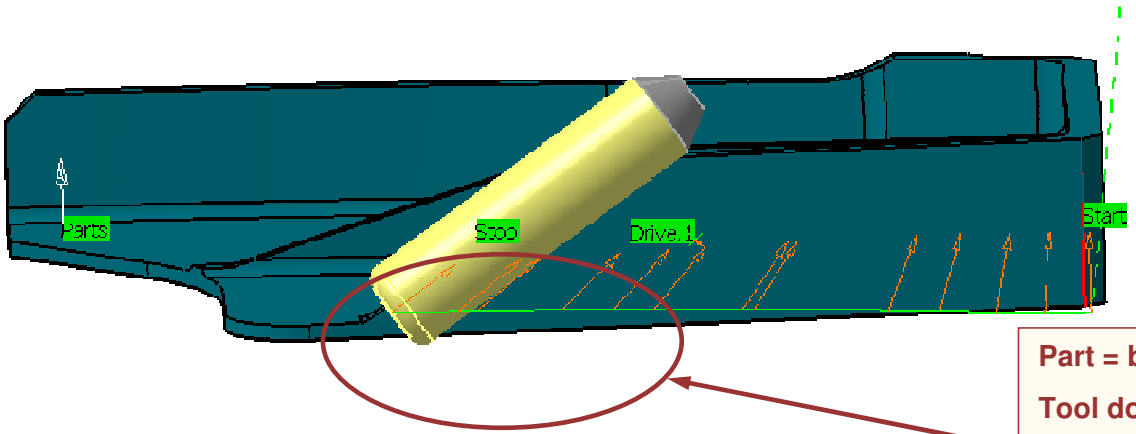
- Use curves as part: example



Part = curve

Tool can cross curve

Better Cutting tool condition



Part = bottom

Tool do not cross bottom

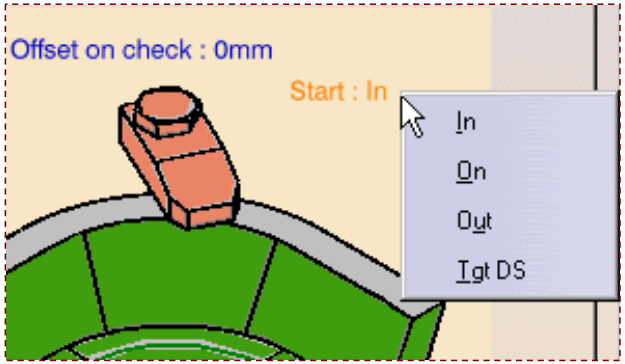
Cutting tool condition not optimized

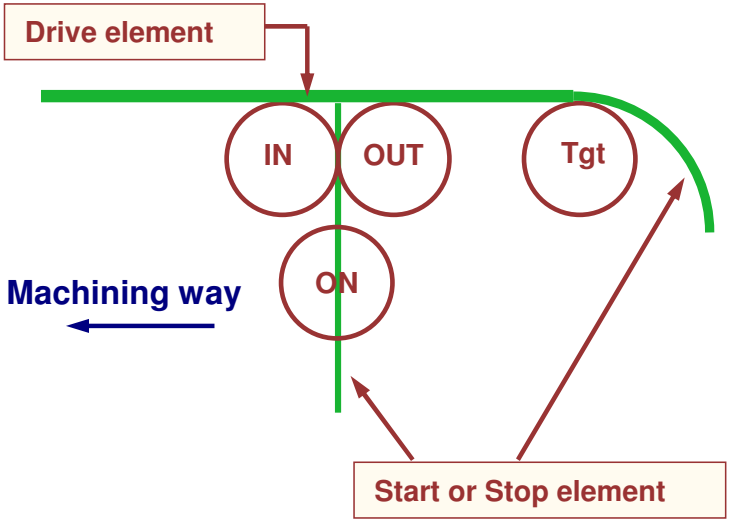
Student Notes:

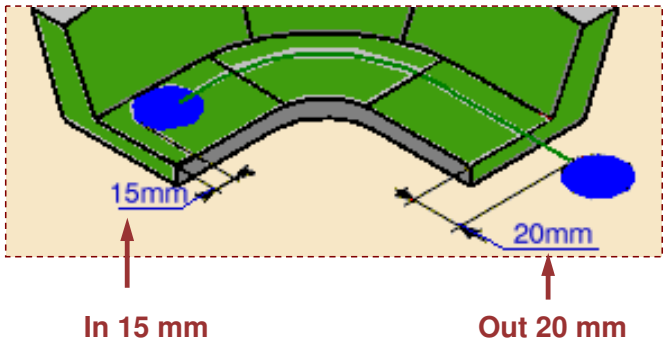
Start and Stop Elements (1/2)

Start and Stop elements must be a surface, a plane, an edge or a vertex.

- **Start element**
  - **The algorithm needs to know a start position. This position is computed using the first selected drive and the start element.**
- **Stop element**
  - **As for the start element, this position is computed using the last selected drive and the stop element.**
- **Start/Stop Conditions**
  - **Positioning of the tool is automatically computed. But it can be modified using right-click on « start » or « stop ».**
  - **An offset can be applied.**





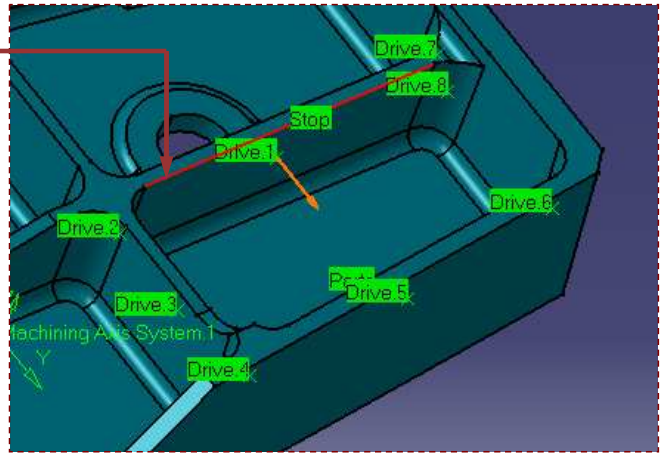


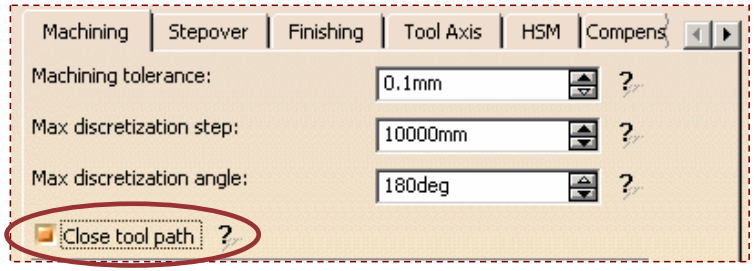
Student Notes:

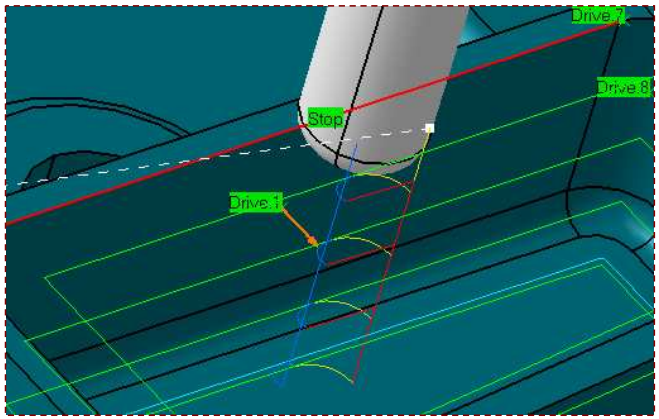
Start and Stop Elements (2/2)

Closed Pocket trick:

- Start and Stop
  - Select top edge from Drive.1 as Start and Stop element (ON option).
  - Algorithm is automatically computing middle point of this edge then creating a virtual plane normal to drive at this point.
  - Thus you can start on Drive 1.
  - Do not forget to activate Close Tool path option.



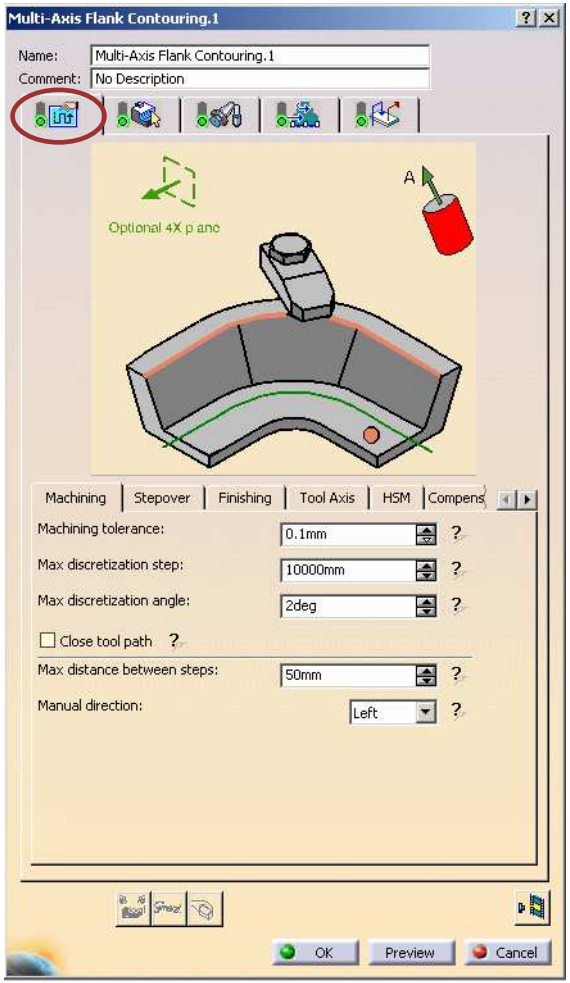




Student Notes:

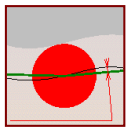
Multi-Axis Flank Contouring: Strategy

You will become familiar with Strategy Tab of Multi-Axis Flank Contouring.

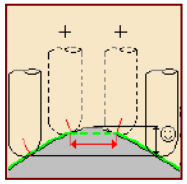


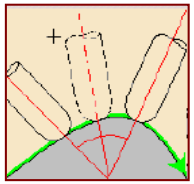
Machining Tab (1/2)

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

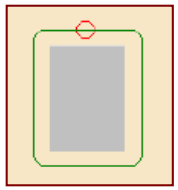


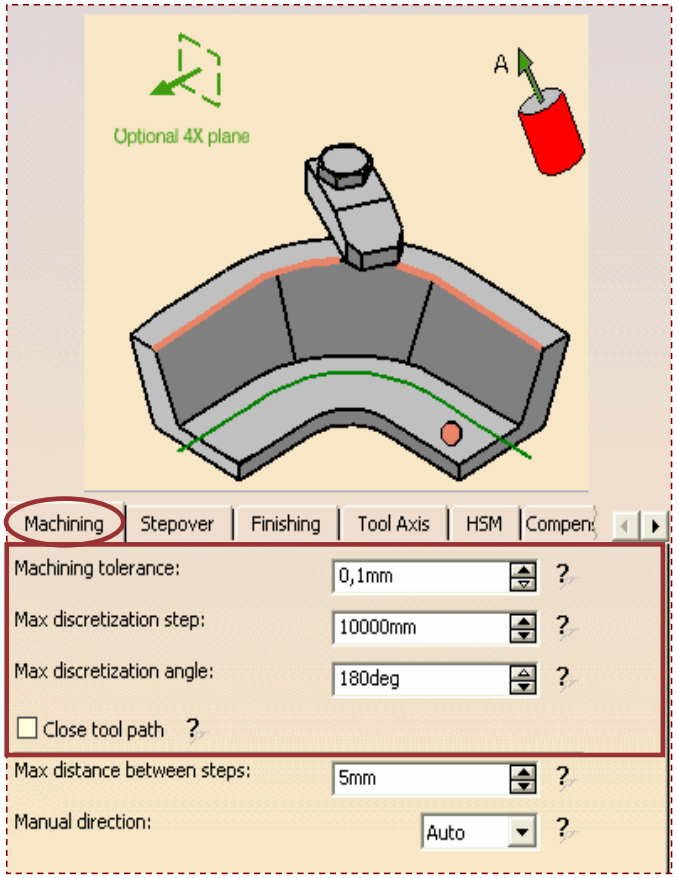
- **Max discretization step and angle**
  - Maximum distance and angle between two outputted points of tool path (default values are infinite, different settings have to be done according to post-processor and machine feature).





- **Close tool path**
  - Option to activate in closed pocket when the first drive element is used as last drive.



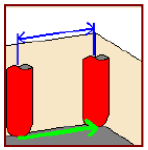


Student Notes:

Student Notes:

Machining Tab (2/2)

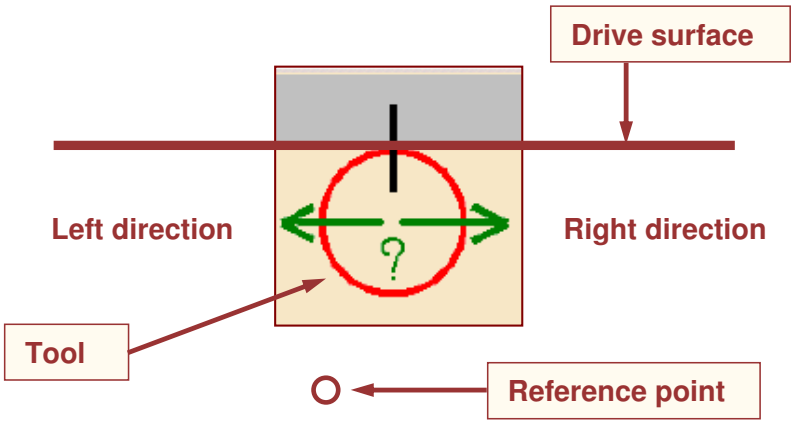
- Maximum distance between steps
  - Rough estimated distance used by the algorithm to search for next drive or check element (In most of cases do not modify this parameter)

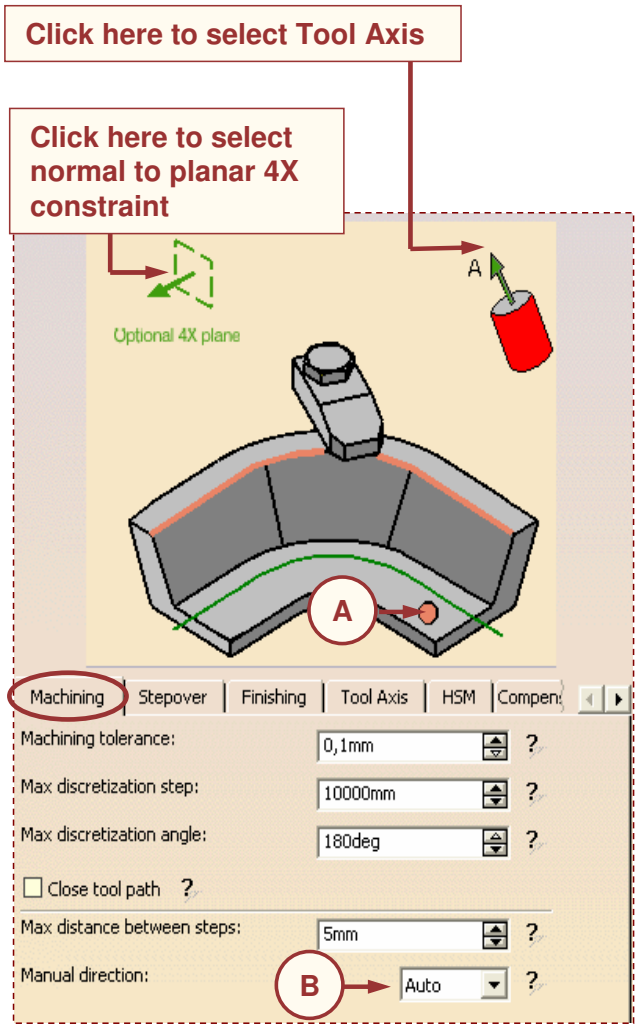


- Reference point and Manual direction

A
- This point is automatically computed (using first drive, part and start element) But in particular geometric cases it could have to be manually defined.

B
- Using a reference point, direction can be automatic, right or left:

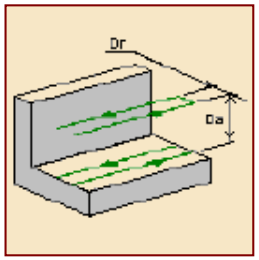




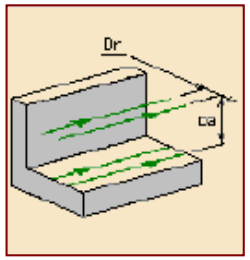
Student Notes:

Stepover Tab (1/2)

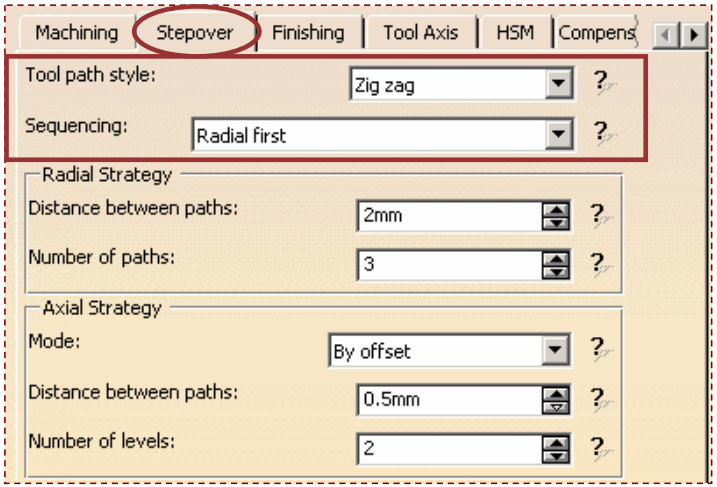
- **Tool path style:**
  - **Zig zag or one way**



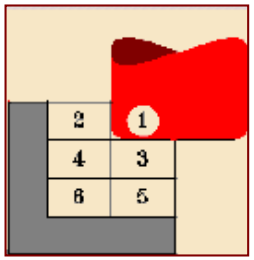
Zig zag



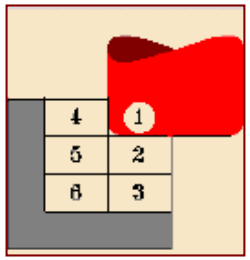
One way



- **Sequencing**
  - **Radial or Axial priority**



Radial priority

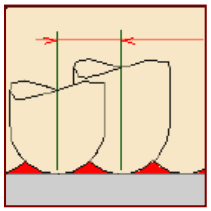


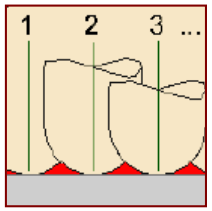
Axial priority

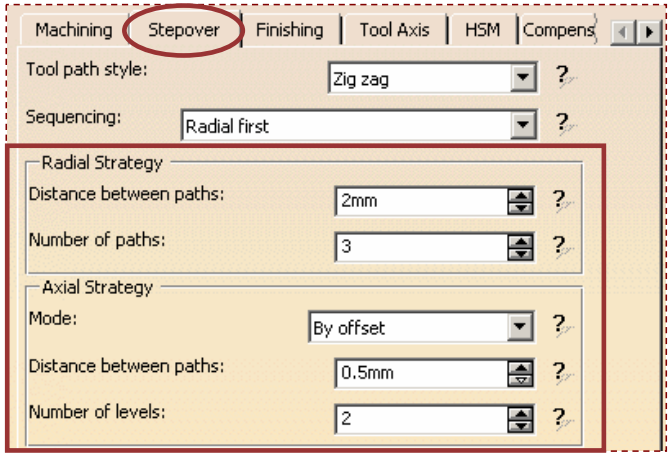
Student Notes:

Stepover Tab (2/2)

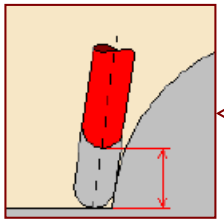
- Radial strategy
  - Define the distance between paths and the number of paths



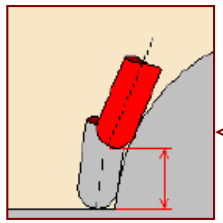




- Axial strategy
  - Select the mode by offset or by thickness

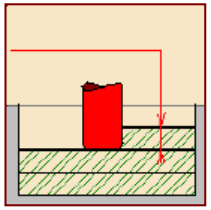


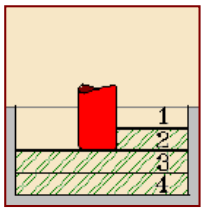
By offset: tool path is computed once then an offset along axis is applied for each level



By thickness: tool path is re-computed for each level

  - Define the distance between paths and the number of levels

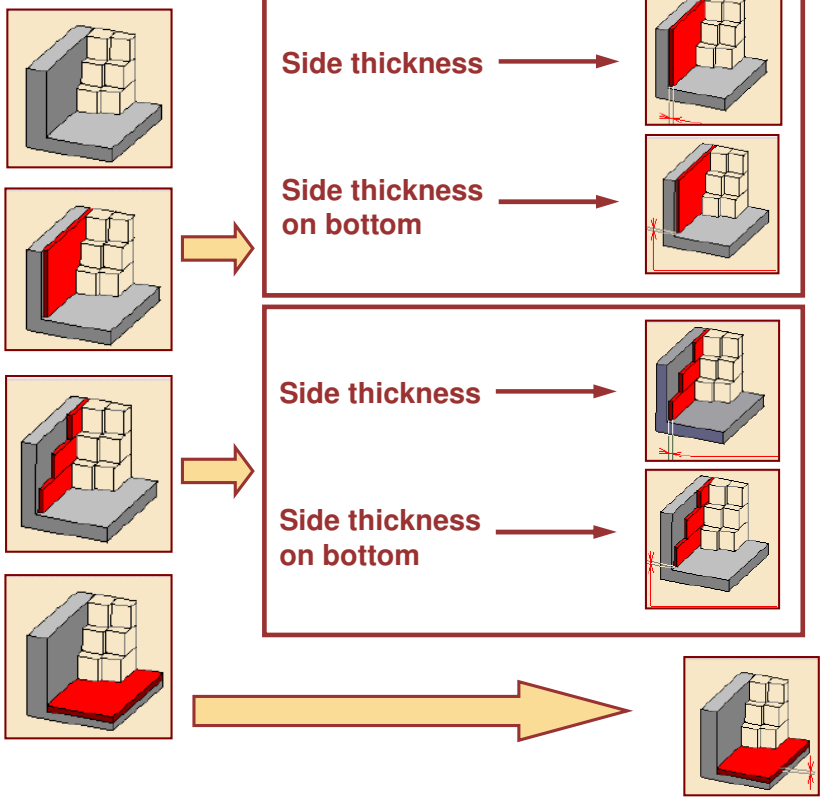


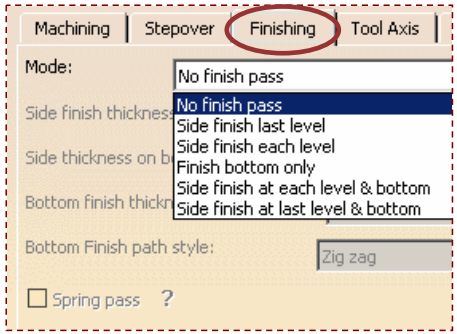


Student Notes:

Finishing Tab (1/2)

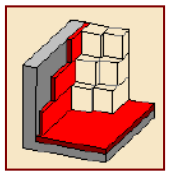
- Mode
  - No side finish
  - At last level
  - At each level
  - At bottom



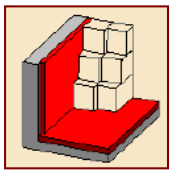


- Each side and bottom finishing strategies can be combined:

At each level and bottom



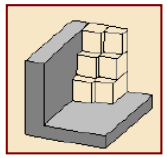
At last level and bottom



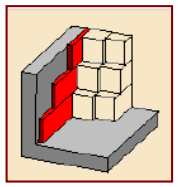
Student Notes:

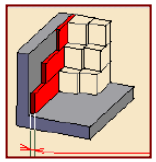
Finishing Tab (2/2)

- Mode
  - No side finish

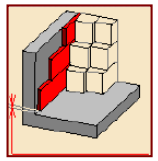


- At each level



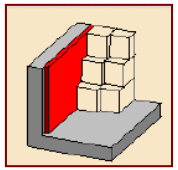


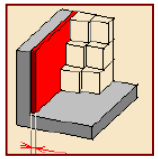
Side thickness



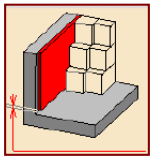
Side thickness on bottom

- At last level



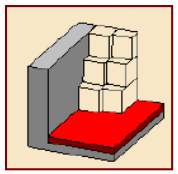


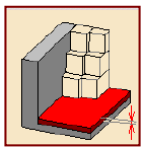
Side thickness



Side thickness on bottom

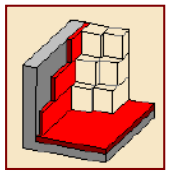
- At bottom



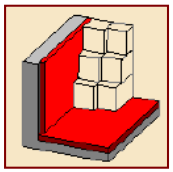


- Each side and bottom finishing strategies can be combined as follows:

  At each level and bottom

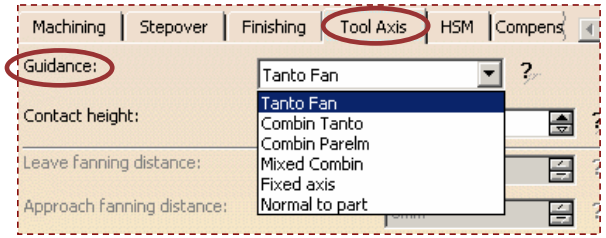


  At last level and bottom



Tool Axis Tab (1/12)

All Guidance Definitions:



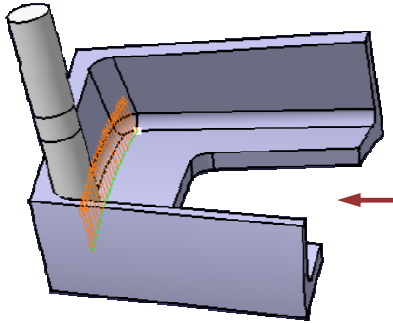
Guidance	Definition
Tanto Fan	This is the basic strategy which ensures good continuity through the different tool motions. It is less used for flank contouring in structural parts as you don't control collision with the drive. To a safety use, add a guiding curve.
Combin Tanto	This strategy ensures that the tool stays normal to the Part in the forward direction with a fanning at the beginning and at the end of the tool motion. This strategy is good for circular and planar drive surfaces where the isoparametric curves are not proper (incline isopararimetric) to force the tool to have a minimum lead angle.
Combin Parelm	This strategy ensures you that the tool will follow the "isoparametrics" of your surface with a fanning at the beginning and at the end of the tool motion. It is the strategy to use when the isoparametric of the drive surfaces have a good orientation.
Mixed Combin	This strategy provides a COMBIN TANTO on planar and cylindrical rsur and a COMBIN PARELM in other cases. It is the better strategy in most of the cases for structural part flank contouring.
Fixed axis	The tool axis is fixed to a defined direction.
Normal to part	The tool stays normal to the part surface.

Student Notes:

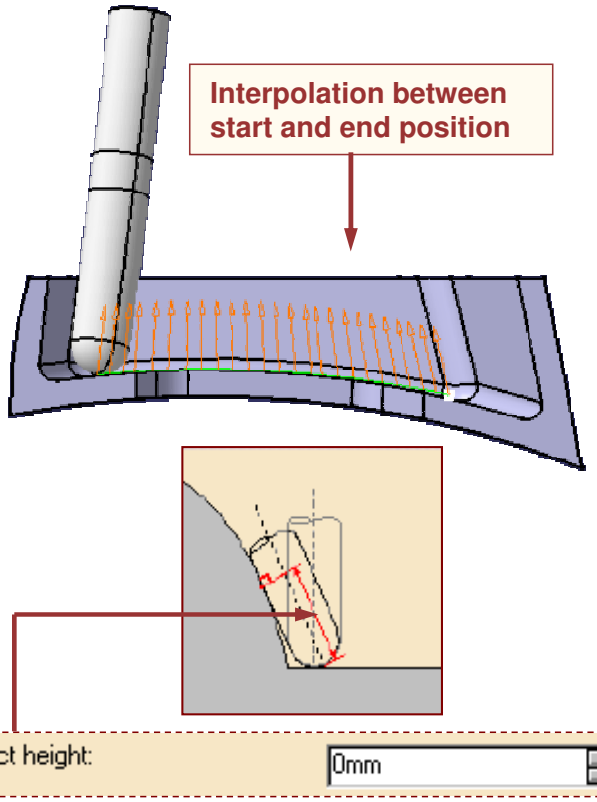
Student Notes:

Tool Axis Tab (2/12)

- Guidance: Tanto Fan
  - Tool is tangent to the drive surface at a given contact height. Tool axis is the interpolation between the start and end positions.



Tool tangent to the drive at a given contact height point.

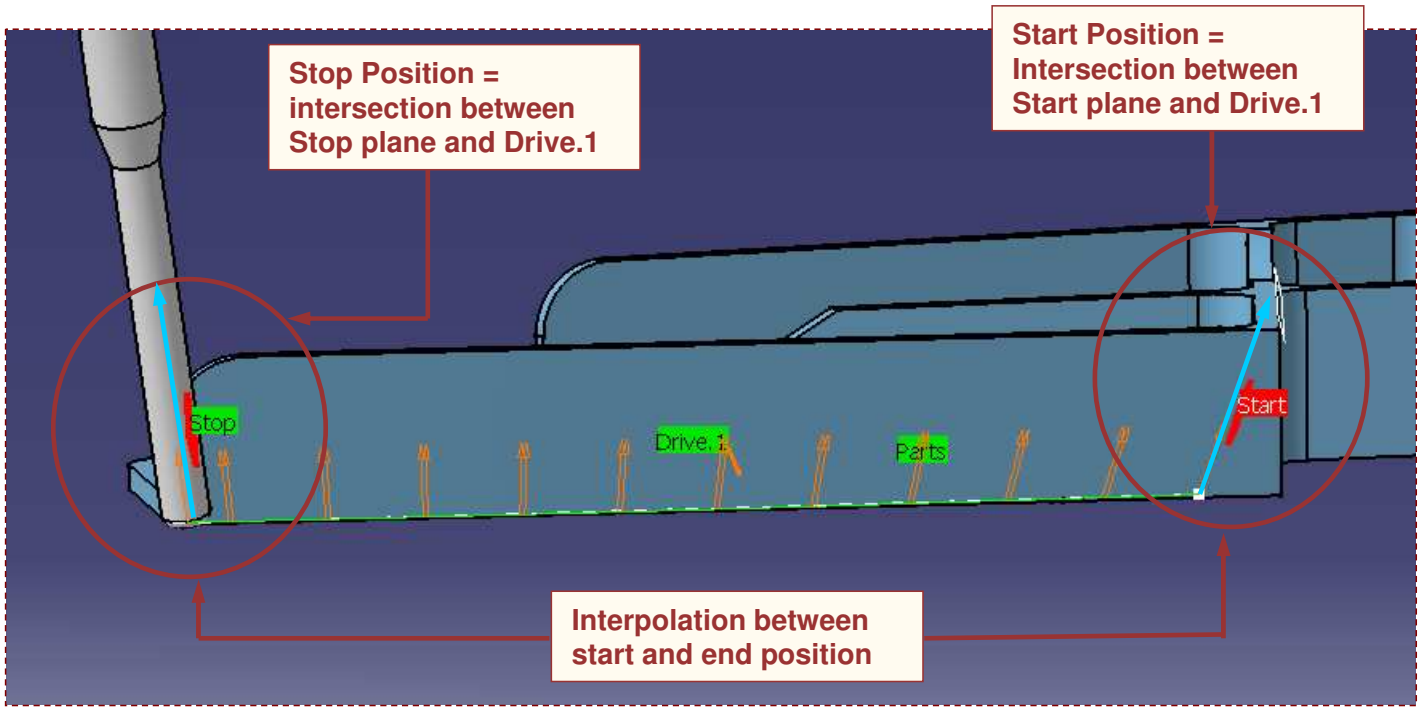


  - The contact height is used to determine a point on the drive surface where the tool must respect tangency conditions. Default value is zero and is related to the bottom of the tool.

Tanto Fan: This is the basic strategy which ensures good continuity through the different tool motions. It is the less used strategy for flank contouring in structural part as you can't control collision with the drive. To a safety, use add a guiding curve.

Tool Axis Tab (3/12)

Tanto Fan Guidance illustration:



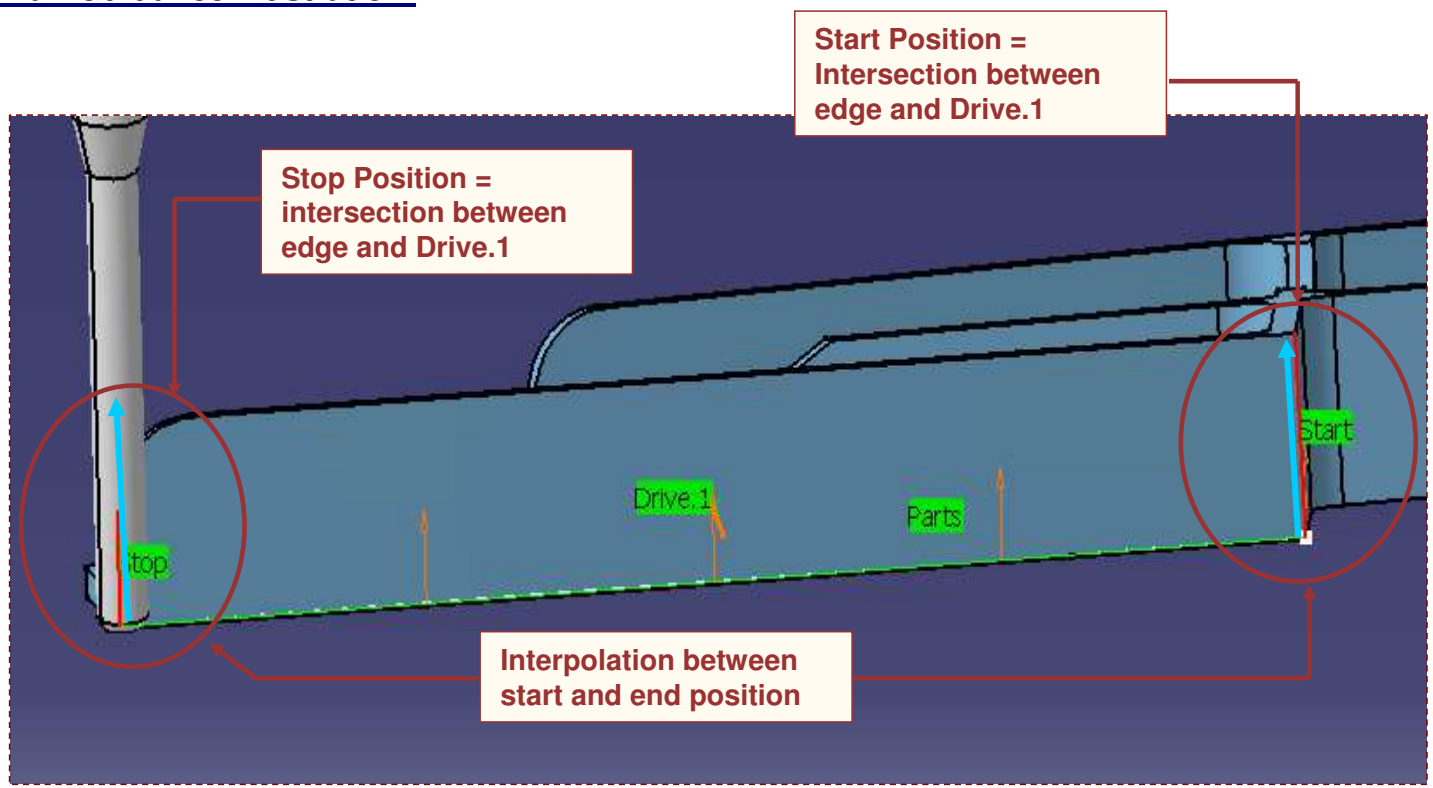
In this example, Start/End Position are defined using planes (manually created).

Student Notes:

Tool Axis Tab (4/12)

Student Notes:

Tanto Fan Guidance illustration:

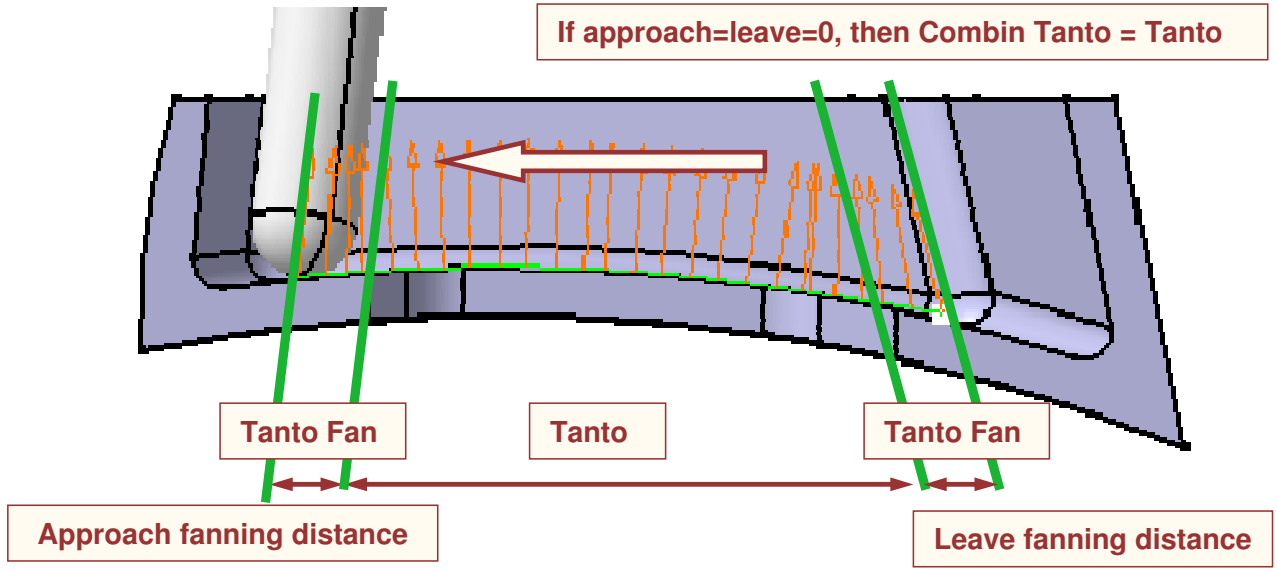


Same example, but Start/End Position are defined using edges of Drive.1

Student Notes:

Tool Axis Tab (5/12)

- Guidance: Combin Tanto
  = Tanto Fan (during leave distance) + Tanto + Tanto Fan (during approach distance)
- Tanto guidance definition: (exists alone only as a local mode)
  - Tool is tangent to the drive surface at a given contact height.
    Tool Axis contained in a plane normal to forward direction
  - Approach and leave distance parameters can be modified

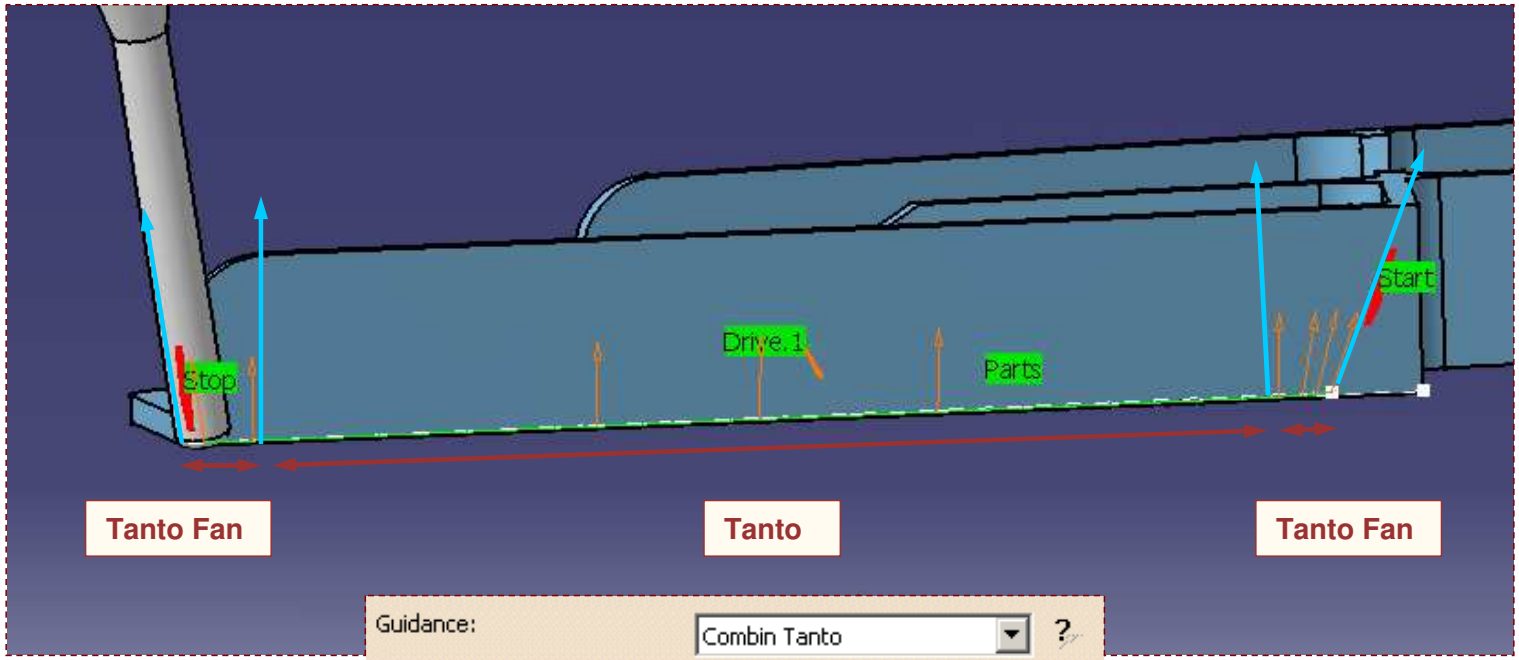


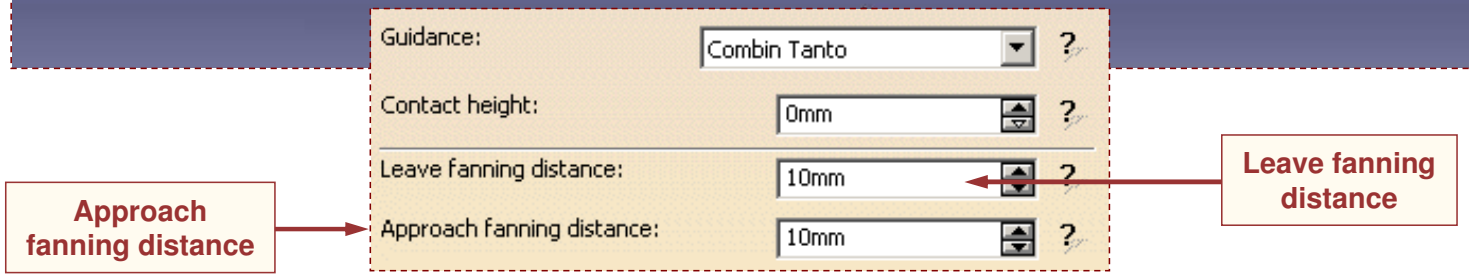
Combin Tanto: This strategy ensures that the tool stays normal to the Part in the forward direction with a fanning at the beginning and at the end of the tool motion.
This strategy is good for circular and planar drive surfaces where the isoparametric curves are not proper (incline isopararimetric) to force the tool to have a minimum lead angle.

Tool Axis Tab (6/12)

Student Notes:

Combin Tanto Guidance illustration:

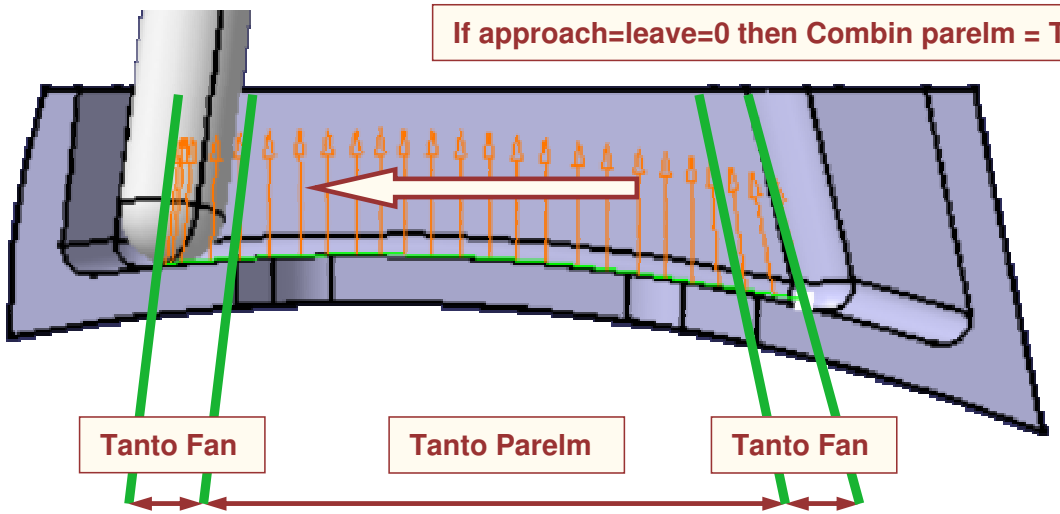




Student Notes:

Tool Axis Tab (7/12)

- Guidance: Combin parelm
  = Tanto Fan (during leave distance) + Tanto Parelm + Tanto Fan (during approach distance)
- Tanto parelm guidance definition:
  - The tool axis is tangent to the drive surface at the specified contact height and follows the isoparametrics of the Rsur
  - Approach and leave distance parameters can be modified:

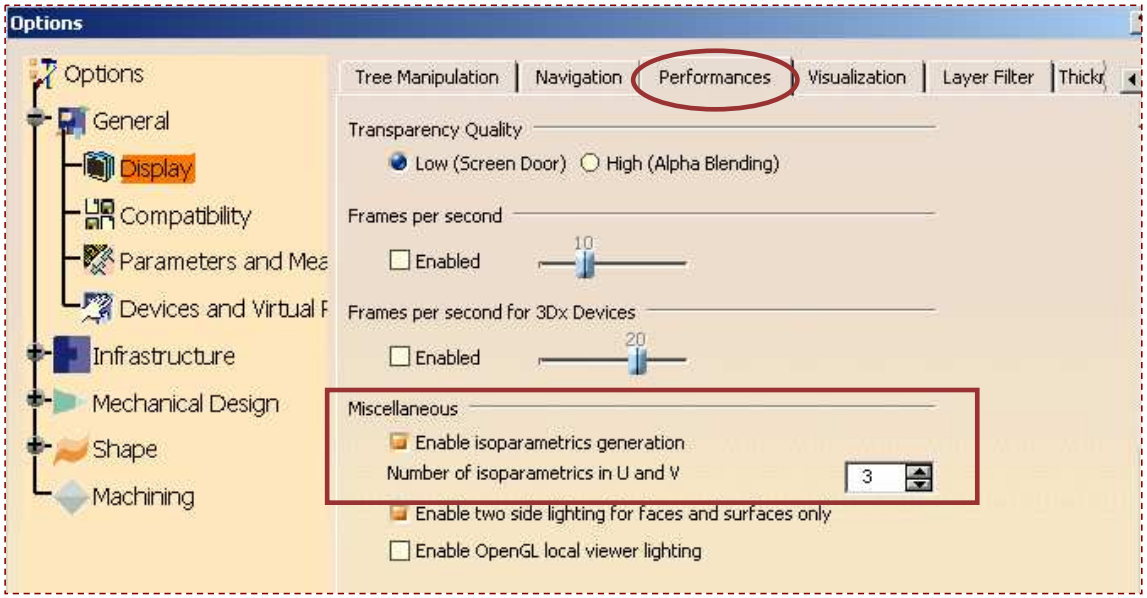


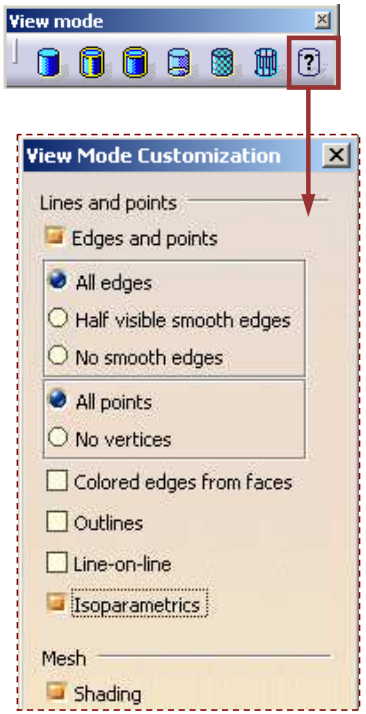
Combin Parelm: This strategy ensures you that the tool will follow the “isoparametrics” of your surface with a fanning at the beginning and at the end of the tool motion.
It is the strategy to use when the isoparametric of the drive surfaces have a good orientation.

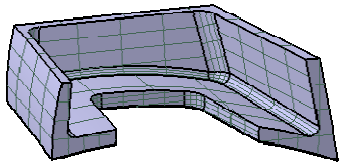
Student Notes:

Tool Axis Tab (8/12)

- Tip: how to see “isopar” in V5
  - Tools/Options/General/Display/Performances/ Enable isoparametrics generation = ON
    - Restart CATIA
  - View/Render Style/Customized View/ Isoparametrics = ON



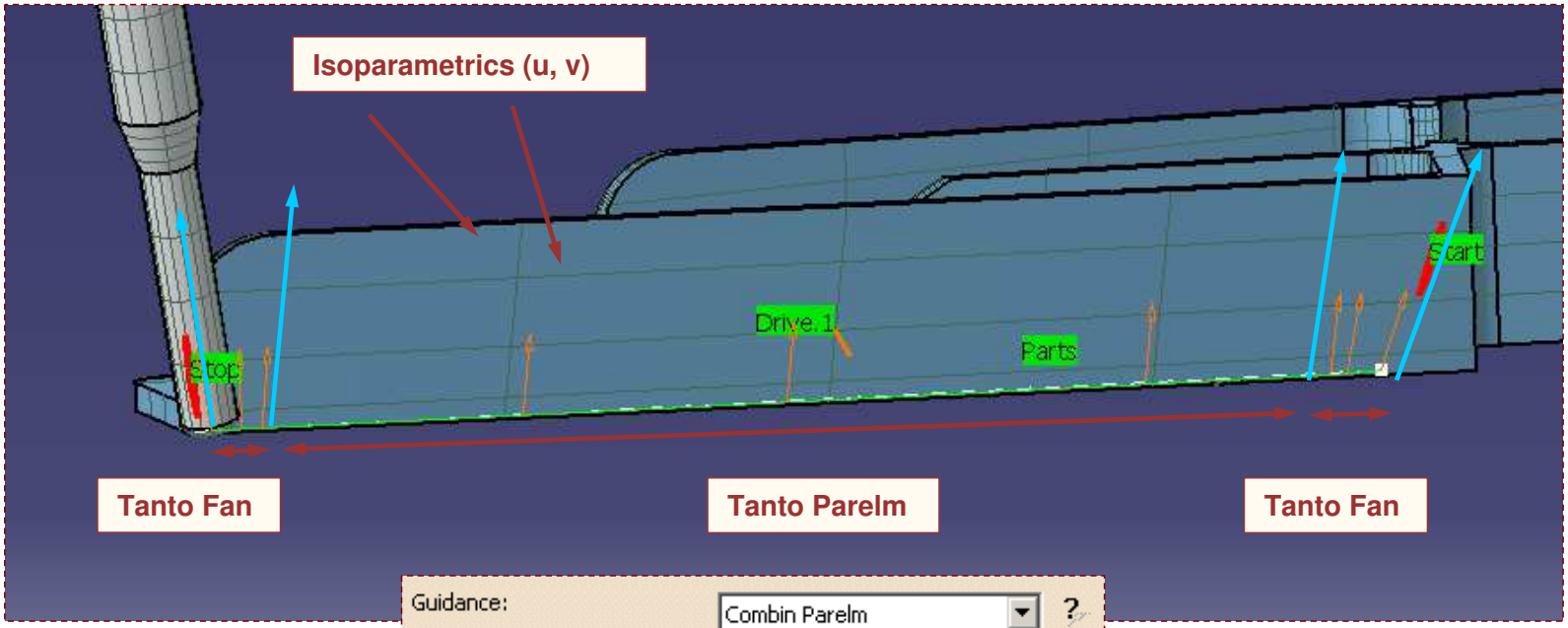


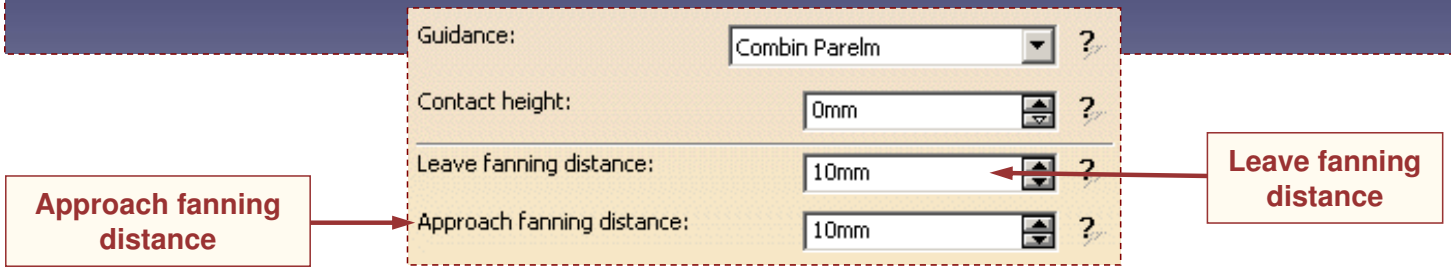


Tool Axis Tab (9/12)

Student Notes:

Combin Parelm Guidance illustration:

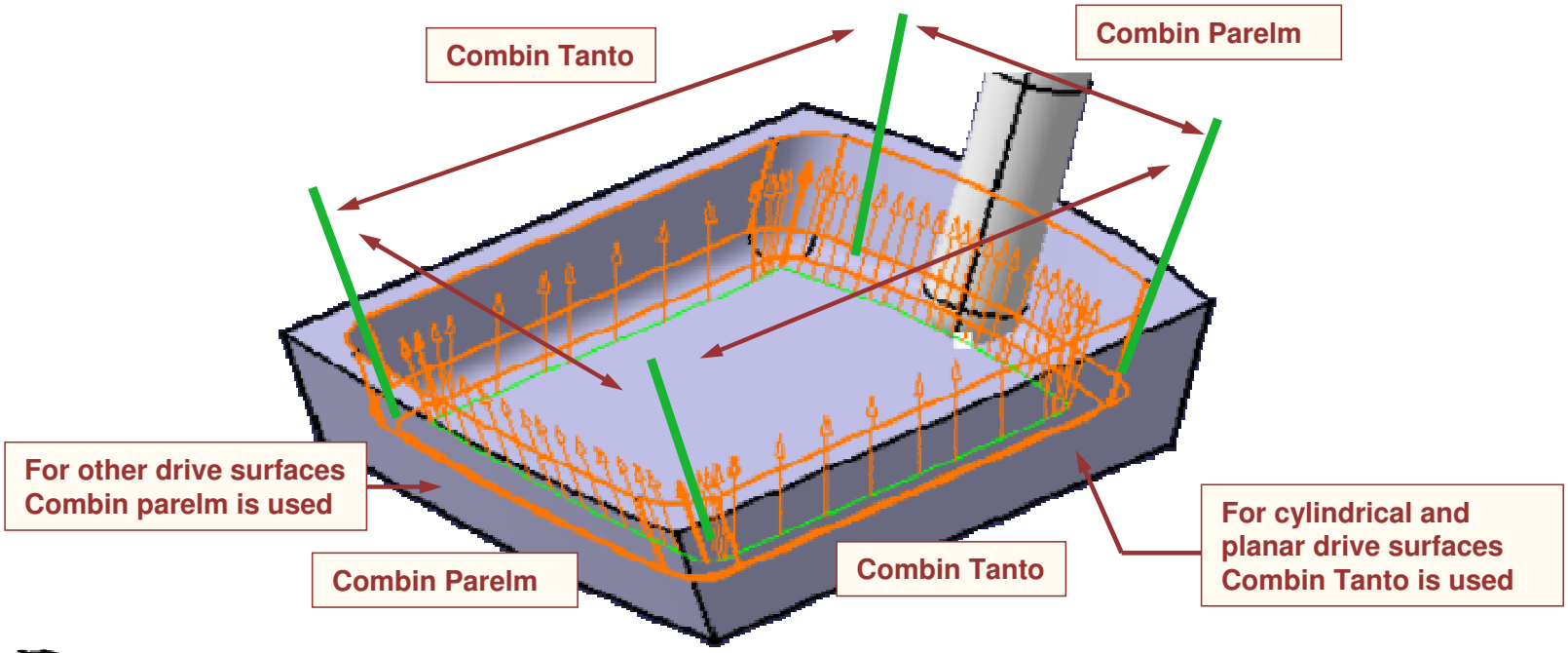




Student Notes:

Tool Axis Tab (10/12)

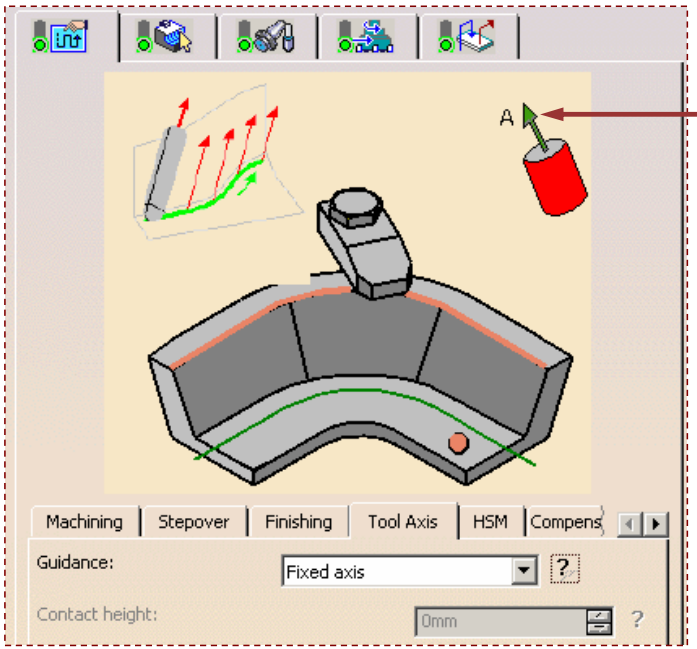
- Guidance: Mixed Combin
  - This strategy is equivalent to Combin Parelm except on planar or cylindrical surfaces on which Combin Tanto strategy will be applied (as isoparametrics direction may not be appropriate to follow on this kind of surface)
  - Approach and leave distance parameters can be modified



Mixed Combin: This strategy provides a COMBIN TANTO on planar and cylindrical rsur and a COMBIN PARELM in other cases.
It is the better strategy in most of the cases for structural part flank contouring.

Tool Axis Tab (11/12)

- Guidance: Fixed Axis
  - Tool Axis is fixed



Click here to select Tool Axis

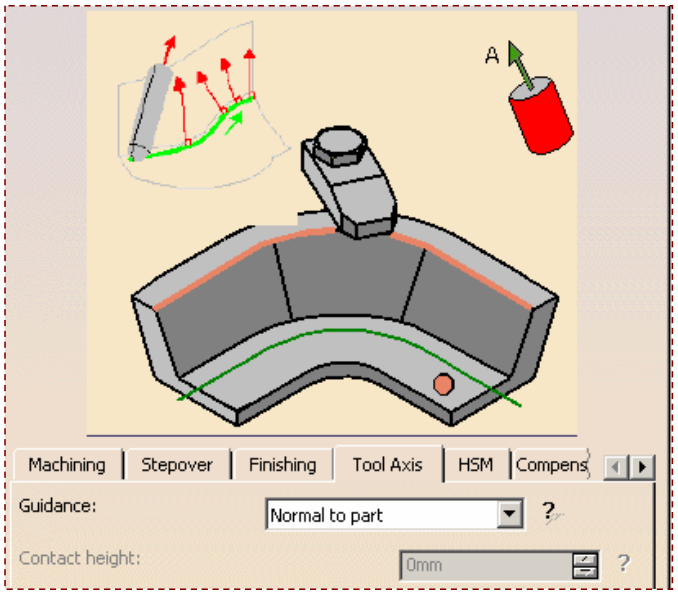
Fixed axis: The tool axis is fixed to a defined direction.

Student Notes:

Student Notes:

Tool Axis Tab (12/12)

- Guidance: Normal to part
  - Tool Axis is normal to selected part while the tool remains in contact with Drives

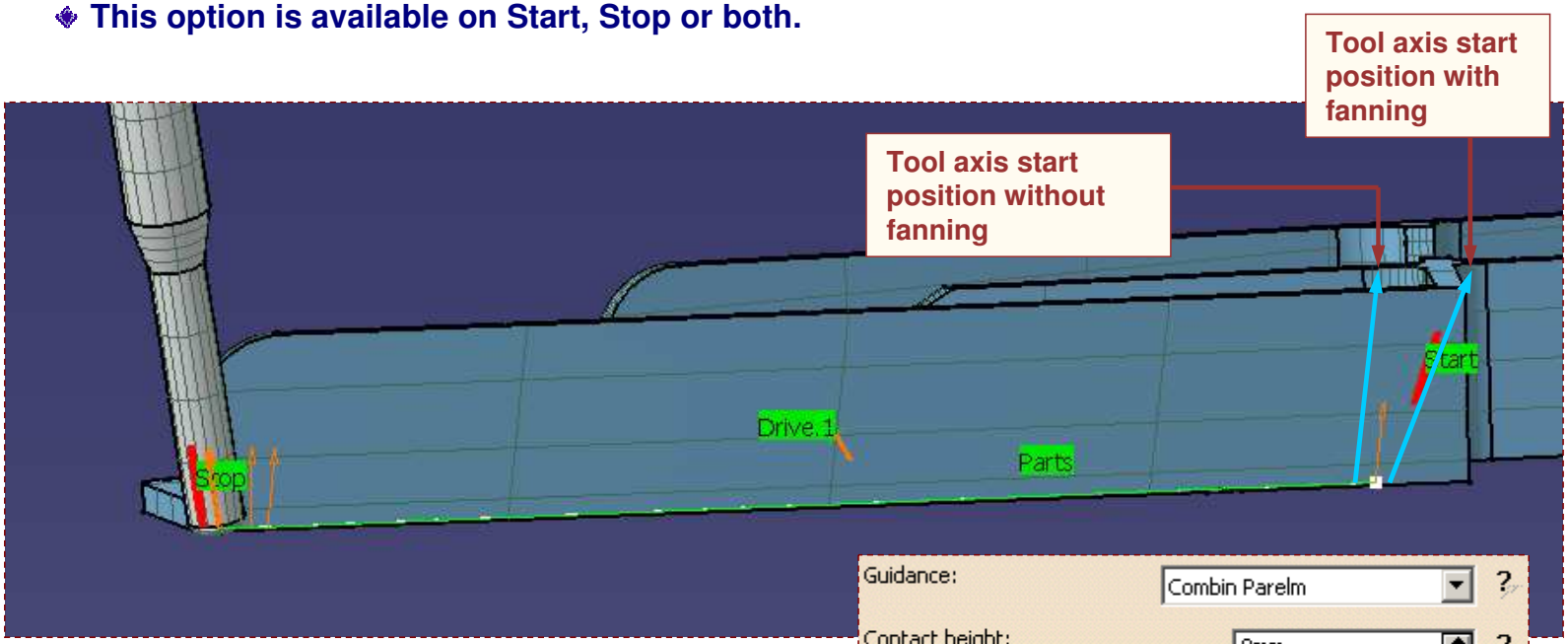


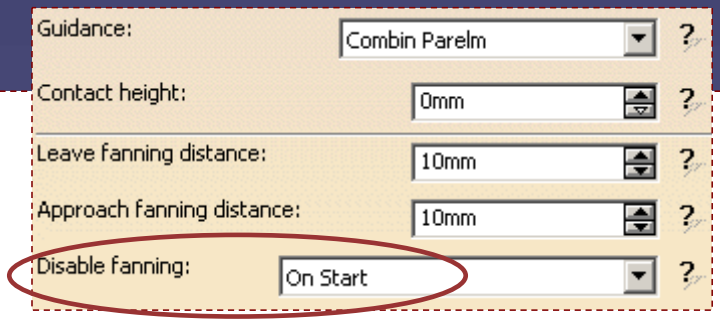
Normal to the part: The tool stays normal to the part surface.

Student Notes:

Other Parameters (1/5)

- Disable fanning:
  - All guidance with fanning (Combin Tanto, Combin parelm and Mixed Combin) can be benefited from disable fanning option.
  - This option is available on Start, Stop or both.



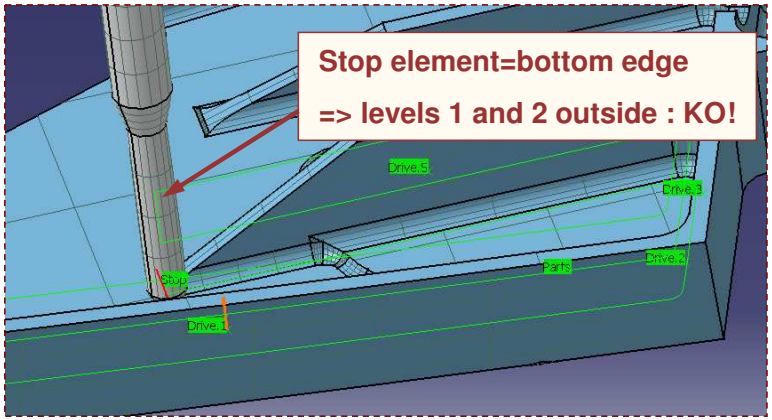


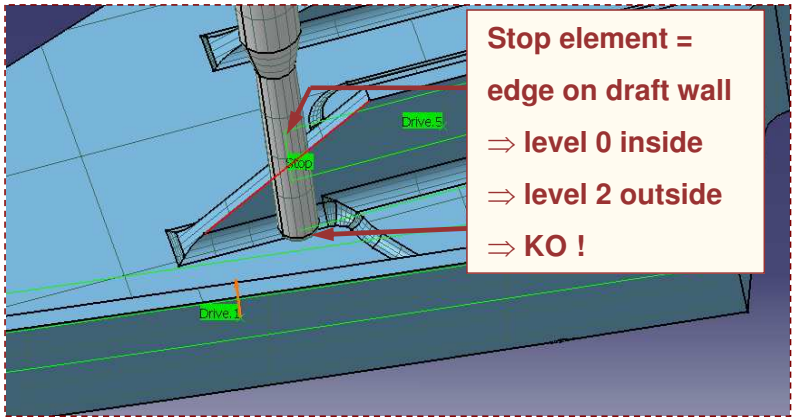
- Disable fanning is an easy way to select start/stop elements connected to machined geometry (edges, rsurs) to keep associativity without unexpected fanning motions.

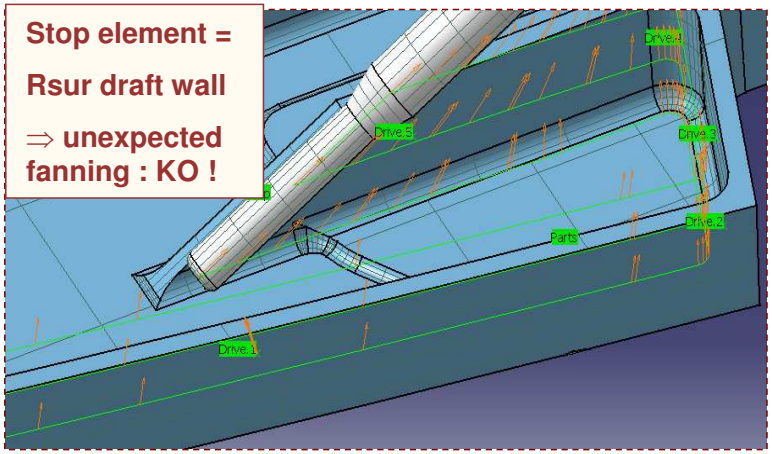
Student Notes:

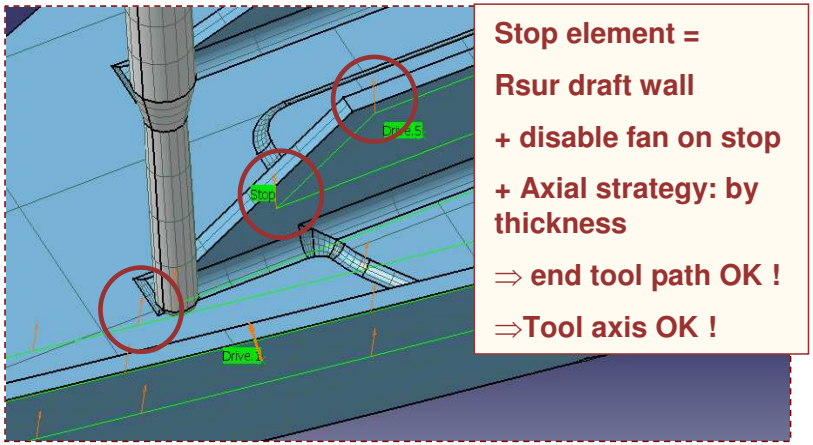
Other Parameters (2/5)

- Disable fanning illustration
  - 5 axis open pocket with draft wall
    - Combin parelm strategy with multi axial levels (0,1, 2)





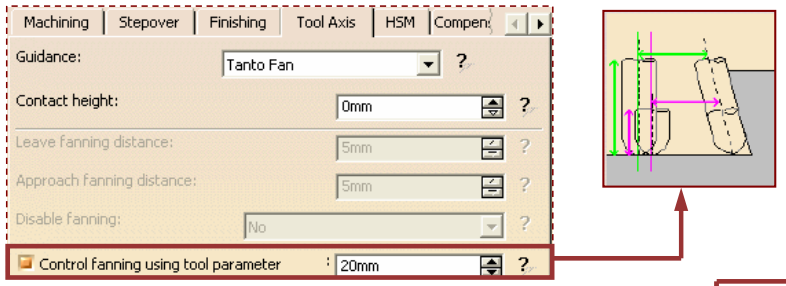


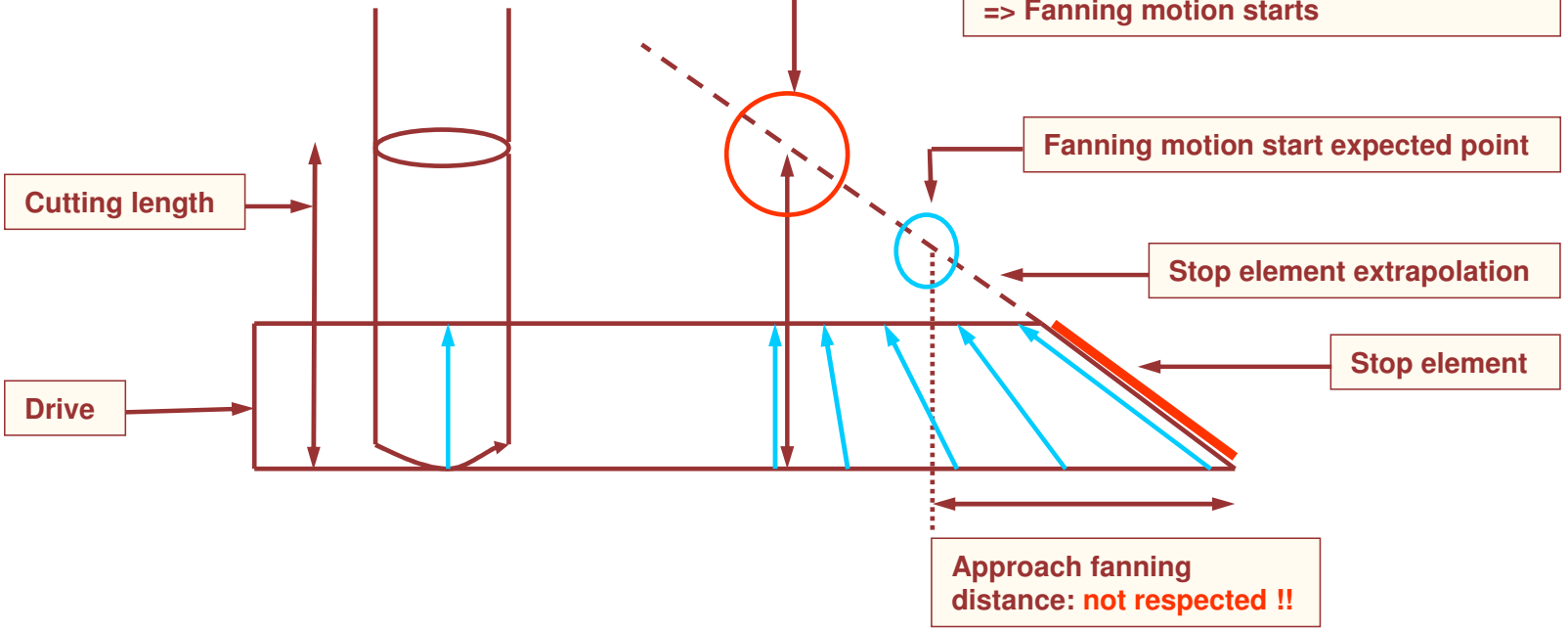


Student Notes:

Other Parameters (3/5)

- Useful cutting length
  - Fanning algorithm is using tool cutting length parameter.
    If needed to control fanning, this parameter can be modified with this option.

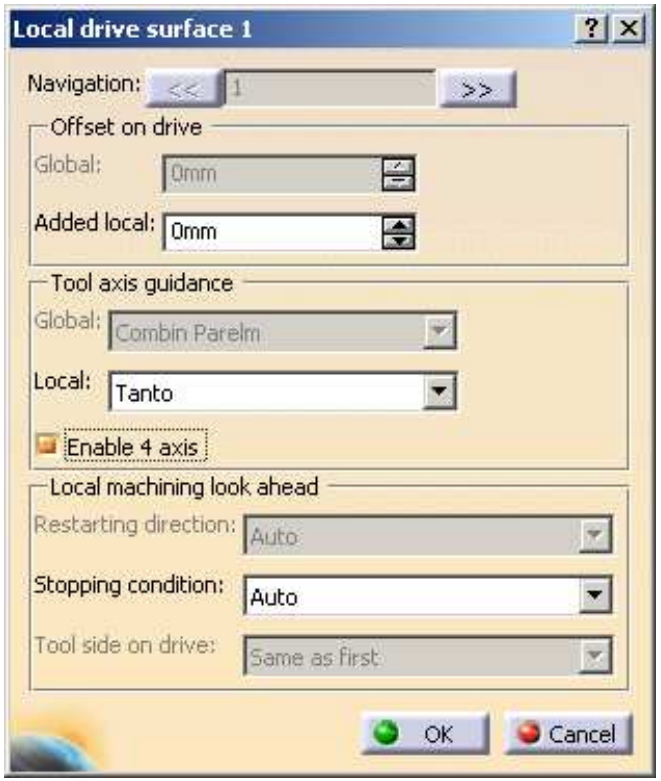


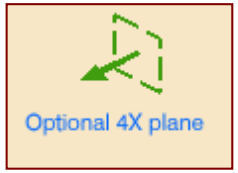


Student Notes:

Other Parameters (4/5)

- Local Tool axis guidance
  - Two axis guidance are available only in local modification.
  - Tanto guidance
    - Tool is tangent to the drive surface at a given contact height. Tool Axis contained in a plane normal to forward direction.
    - In the example, Global guidance is Combin parelm, which means Tanto Parelm on drives (except fanning motions). Then on this, drive guidance will be Tanto.
    - Thus, it is possible to enable 4 axis mode and force Tool axis to be in a manual 4 axis Plane.

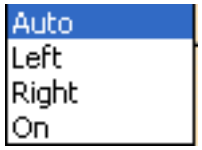


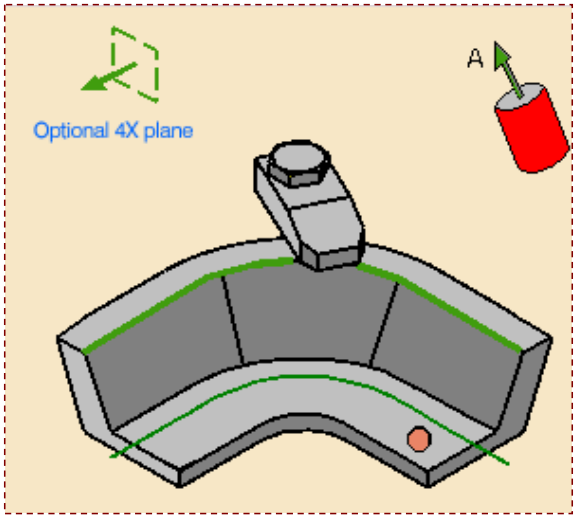


Student Notes:

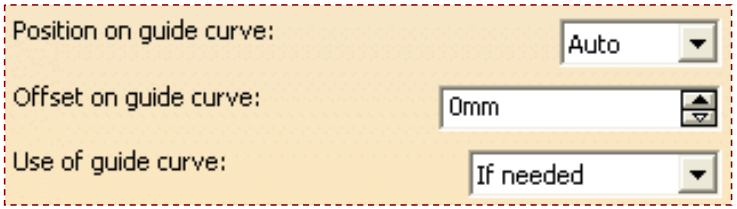
Other Parameters (5/5)

- Auxiliary guiding element
  - This curve is a tool axis strategy modifier. This modifier is used to modify tool axis to avoid collision with Drive Surface.
  - It moves the tool away in the correct direction according to position on guide curve parameter.
    - Auto, Left, Right, On values





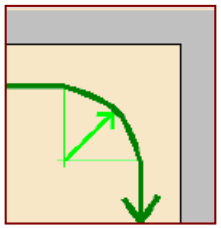
  - It is possible to define offset on guide curve.
  - Always use will force algorithm to always use guide curve.
  - If needed, algorithm will use guide curve only in case of collision.

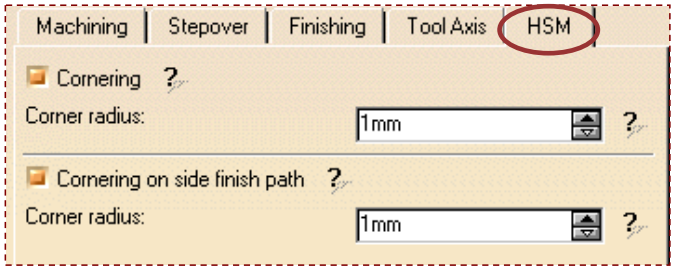


Student Notes:

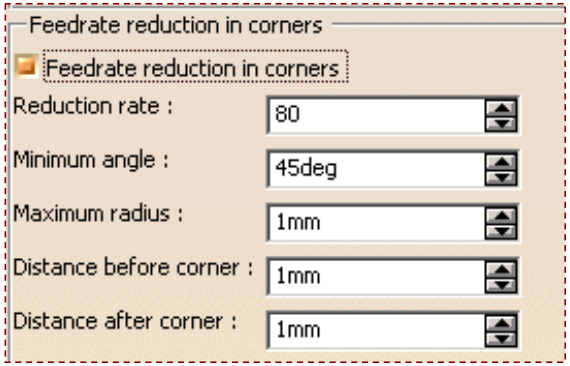
HSM Tab

- Cornering and cornering on side finish path
  - Allow the user to define a cornerisation of the tool path by giving a corner radius





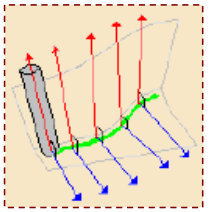
- Feed and Speed Tab 
  - Feed- rate reduction in corners
    - Applied inside corners for machining and finishing passes.
    - Not in macros or default linking and return motions



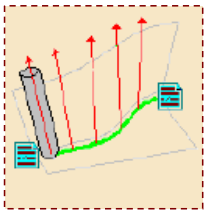
Student Notes:

Compensation (1/2)

- Cutter compensation parameters: Allow you to manage generation of cutter compensation (CUTCOM) instruction in the NC data output
  - 3D radial (PQR):
    3D radial compensation data (Vector <P,Q,R>) can be generated in the APT output. The Radial compensation data output can be activated or not on each Multi Axis Flank Contouring cycle of a program. For all tool positions of the machining passes, for the last motion of each approach macro, and for the first motion of each retract macro, the <PQR> vector is added to the APT statement (which contains the Tip position and the Tool Axis.
    Before the first position with <PQR> data, two APT statements: CUTCOM/SAME,NORMDS and CUTCOM/NORMDS are automatically added, after the last position with <PQR> data, a CUTCOM/OFF statement is automatically added. These PQR statements are supported and can be translated by Multi-Axis Post Processors provided NC manufacturing Workbenches.



  - 2D Radial- TIP (G41/G42) :
    The 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.

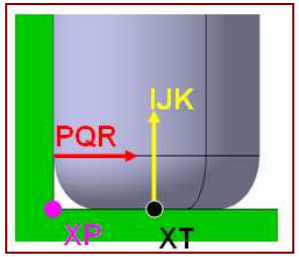


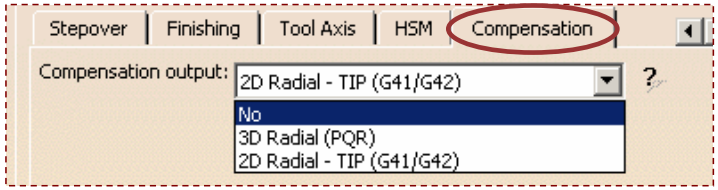
  - None:
    Cutter compensation instructions are not automatically generated in the NC data output. However, CUTCOM instructions can be inserted manually.

Student Notes:

Compensation (2/2)

Compensation output sum up





Internal name	UI name		sub-type	Multax	Output											CUTCOM	NC_Command
Planar	2D Radial	G41/ G42	TIP	Off	XT	YT	ZT									RIGHT or LEFT	NC_CUTCOM_RIGHT / LEFT
			Profile		XP	YP	ZP									OFF	NC_CUTCOM_OFF
Norm_DS	3D Radial	PQR	TIP	On	XT	YT	ZT	I	J	K	P	Q	R			NORMDS	NC_CUTCOM_NORMDS_ON
				Off	XT	YT	ZT	P	Q	R						OFF	NC_CUTCOM_NORMDS_OFF

```
$$  Start generation of : Multi Axis Flank Contouring.1
FEDRAT/ 1000.0000,MMPM
SPINDL/   70.0000,RPM,CLW
CUTCOM/NORMDS
$$ START CUTCOM NORMDS XT,YT,ZT,I,J,K,P,Q,R
GOTO  /  100.00000, -125.00000,   10.00000, 0.000000, 0.000000, 1.000000, $
0.000000, 1.0000000, 0.000000
GOTO  /    0.00000, -125.00000,   10.00000 0.000000, 0.000000, 1.000000, $
0.000000, 1.0000000, 0.000000
CUTCOM/OFF
$$ END CUTCOM NORMDS XT,YT,ZT,I,J,K,P,Q,R
$$  End of generation of : Multi Axis Flank Contouring.1
```

Student Notes:

To Sum Up

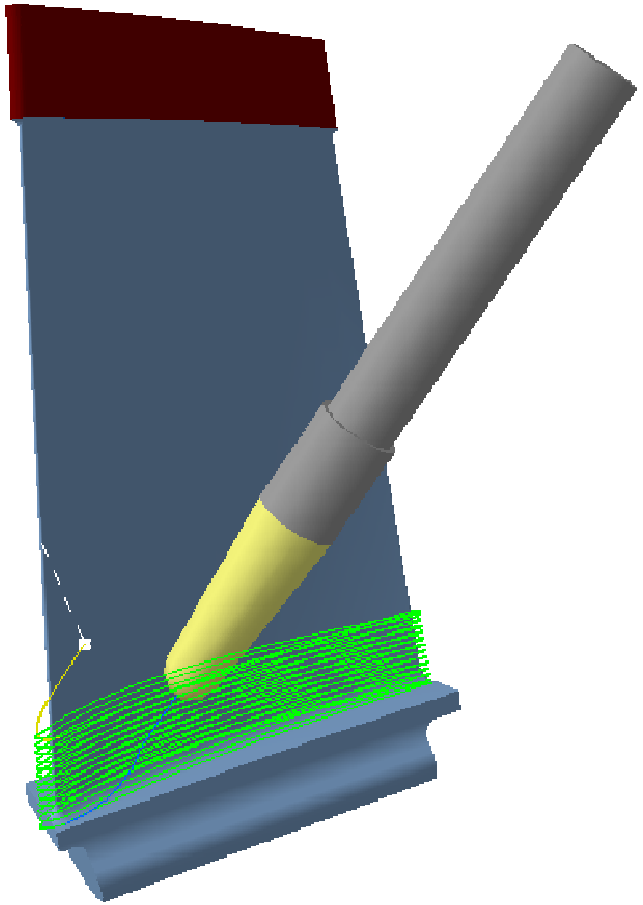
In this course you have seen:

- Necessary geometrical elements to define a Flank Contouring operation
  - Drives
    - Navigation on drives, local modification on drives, non- contiguous drives
  - Parts (can be a curve)
    - Multi part
  - Start/Stop
    - Open or closed pocket
- 5 Axis strategies of Flank Contouring operation
  - Tanto Fan, Combin Tanto, Combin Parelm, Mixed Combin, Fixed axis, Normal to part, 4-Axis
- Stepover management
  - Multi-radial
  - Multi-axial with thickness or offset
  - Side and bottom finishing strategies
- HSM option
- Output: Compensation

Multi-Axis Helix Machining

You will become familiar with the 5 axis Helix Machining principles.

- Introduction
- General Process
- Geometry
- Strategy
- To sum Up



Student Notes:

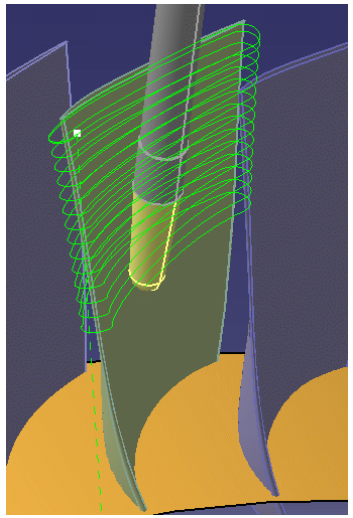
About Multi-Axis Helix Machining

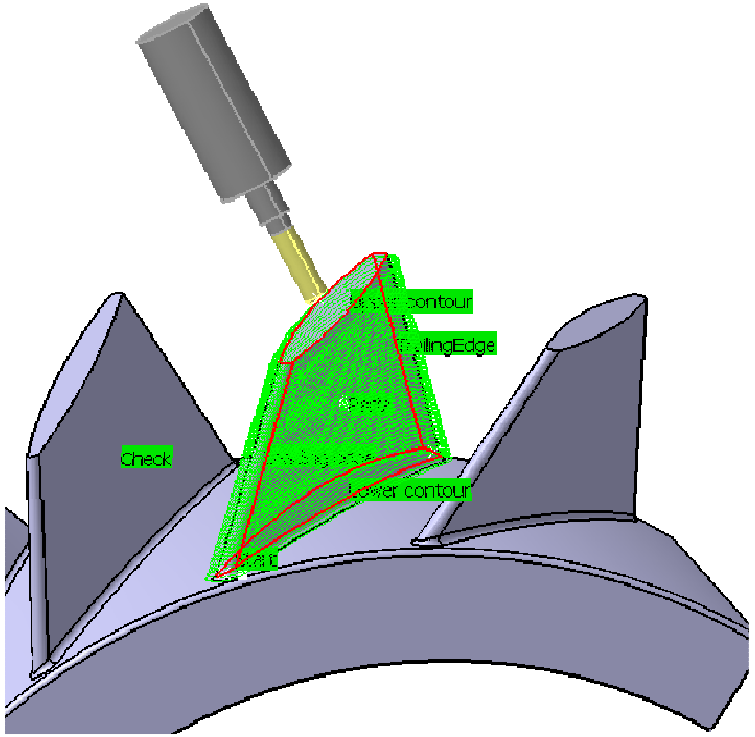
Multi-Axis Helix Machining operation is mainly used for semi-finishing and finishing of blades and blisks in turbo-machinery parts.

Collision checking is possible on cutting part of the tool or on the tool assembly.

Tool Axes can be defined manually in order to have better control on tool and collision can be avoided with neighboring blades.

You can use Multi-Axis Helix Machining operation to generate a single helix toolpath to mill an entire turbo-machinery blade.





Student Notes:

Student Notes:

Multi-Axis Helix Machining: 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 text of comment (optional).

3. Define operation parameters using the 5 tab pages.

Strategy tab

Geometry tab

Tool Definition tab

Feeds & Speeds tab

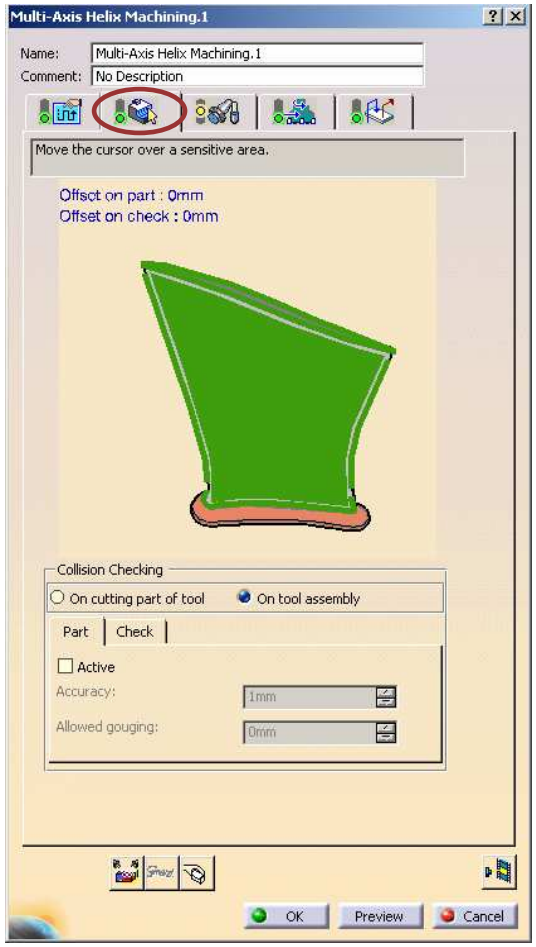
Transition Paths tab

4. Replay and/or Simulate the tool path.

Student Notes:

Multi-Axis Helix Machining: Geometry

You will become familiar with the options on the Geometry tab of Multi-Axis Helix Machining.

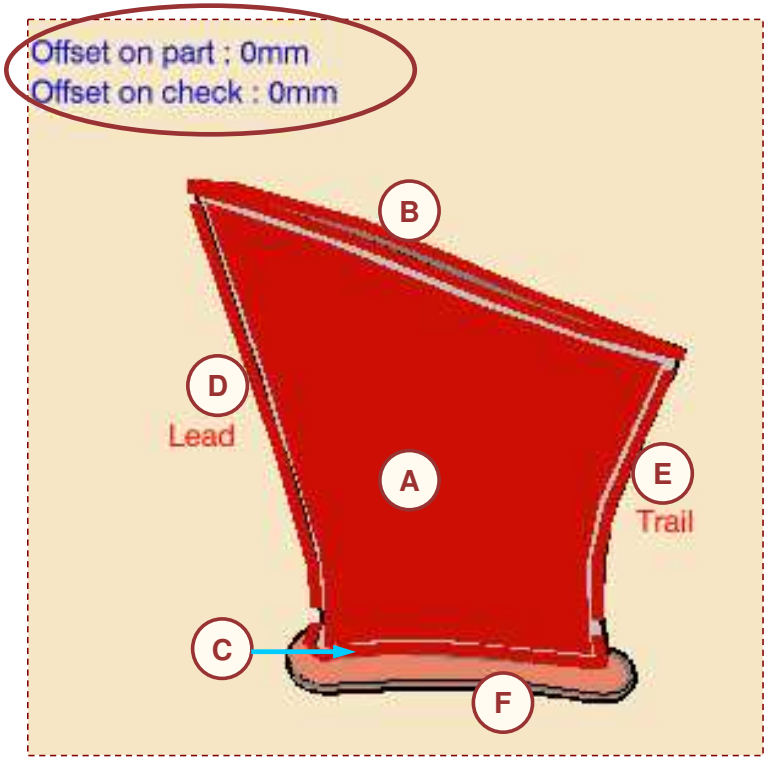


Student Notes:

Presentation

The Geometry tab includes a sensitive icon dialog box that allows the selection of:

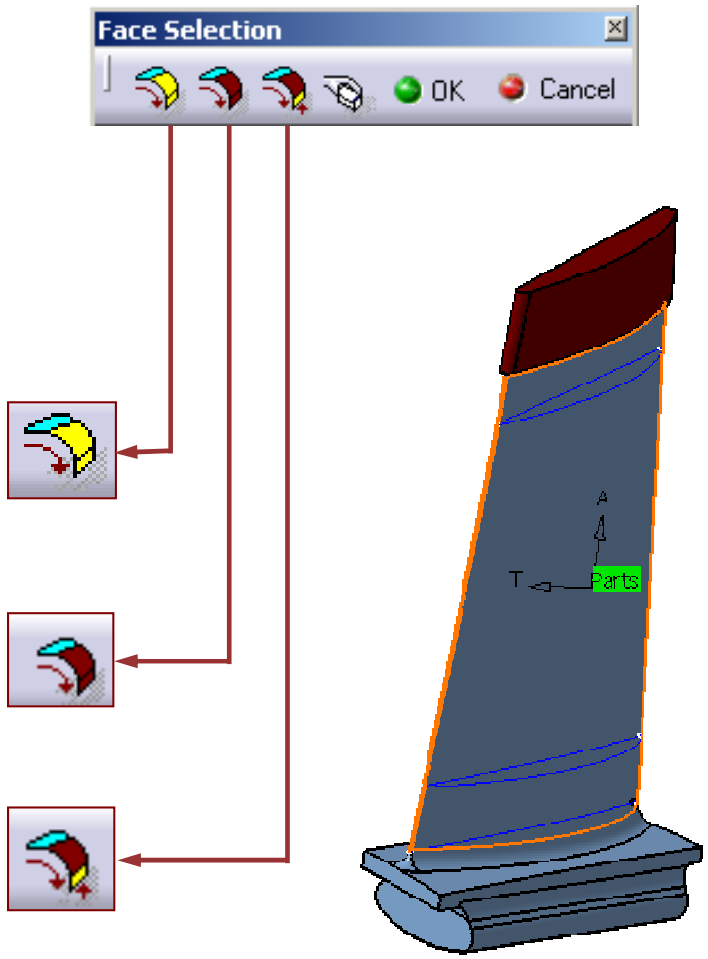
- **Part elements** (A)
  - **blade body surfaces**
- **Upper contour** (B)
  - **limiting element on top of blade**
- **Lower contour** (C)
  - **limiting element on bottom of blade**
- **Leading edge** (D)
- **Trailing edge** (E)
- **Check element (optional)** (F)
  - **elements to avoid during tool path**
- **Offset can be applied on both, part and check.**



Student Notes:

Part Elements

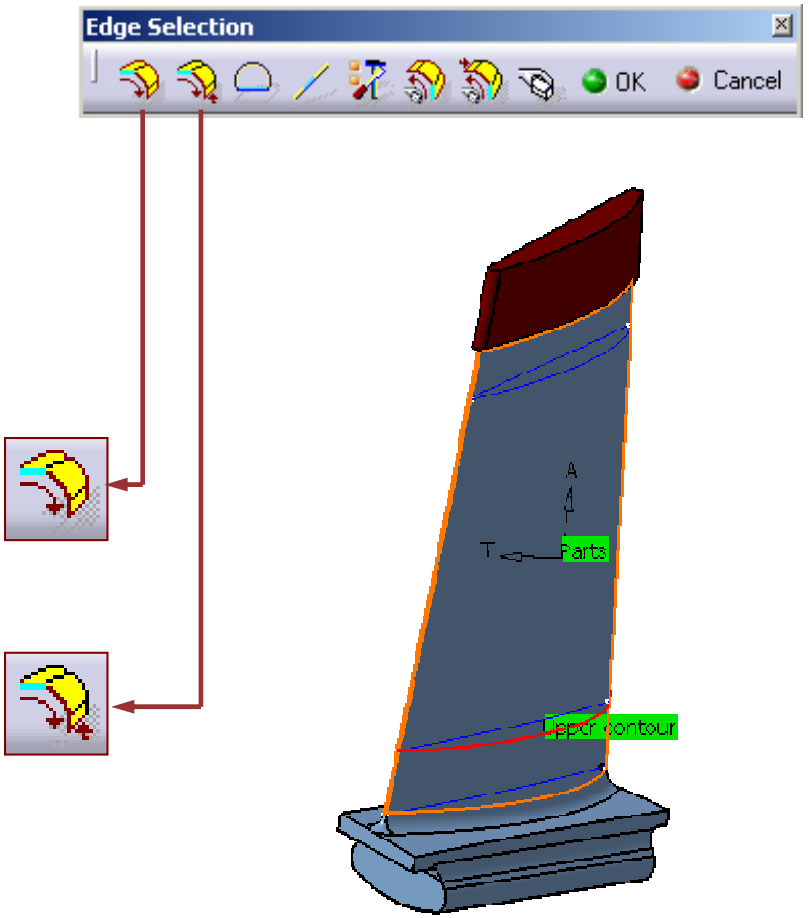
- **Face selection:**
  This wizard allows to select quickly part elements
  - **To start the navigation, one always need to select at least two faces (first one is start element, second one give the direction to navigate).**
  - **Then select**
    Navigates on faces
    Navigation is done on all adjacent faces.
    or
  - **Navigates on belt of faces**
    Navigation is done in order to follow a belt or
  - **Navigates on Faces Until a Face**
    Navigation is done until a selected face.



Student Notes:

Upper and Lower Contours (1/2)

- **Edge selection:**
  This wizard allows to select quickly contour elements

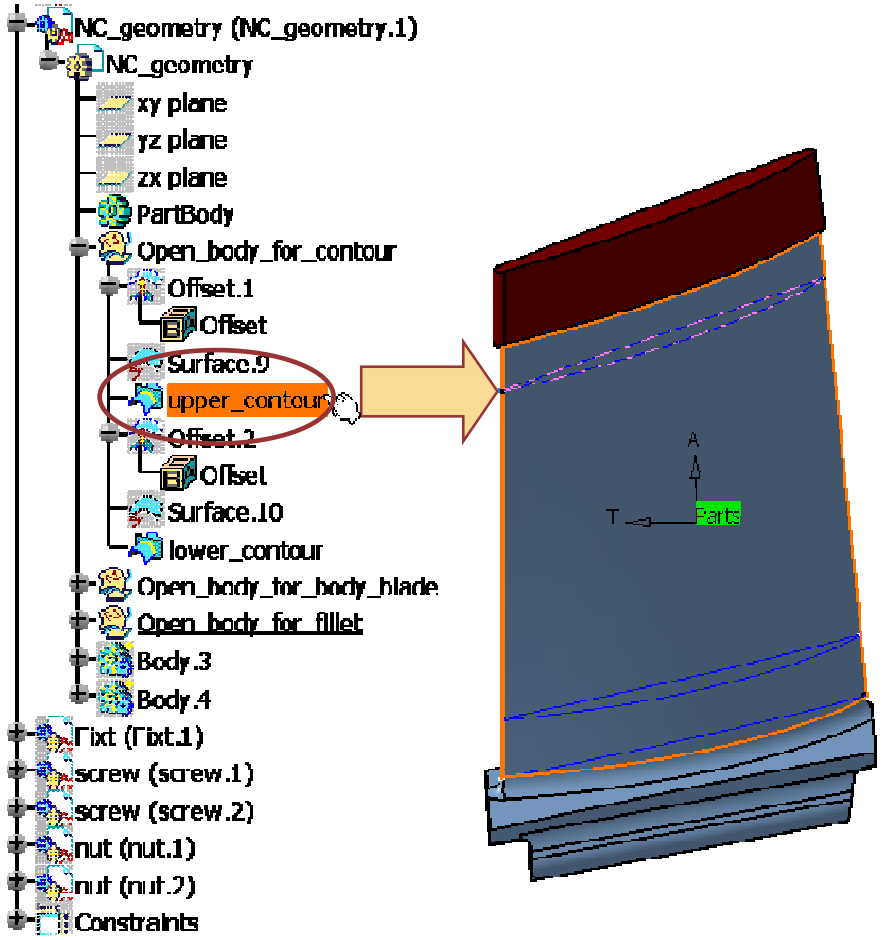


- **To start the navigation, one always need to select at least two edges (first one is start element, second one give the direction to navigate).**
- **Then one can select**
  Navigates on belt of edges
  Navigation is done in order to follow a belt.
  or
- **Navigates on Edges Until an Edge**
  Navigation is done until a selected edge.

Student Notes:

Upper and Lower Contours (2/2)

- Upper and lower contour may be already prepared in NC geometry CATPart.
  - NC programmer will just have to select it in PPR Tree or in CATIA window



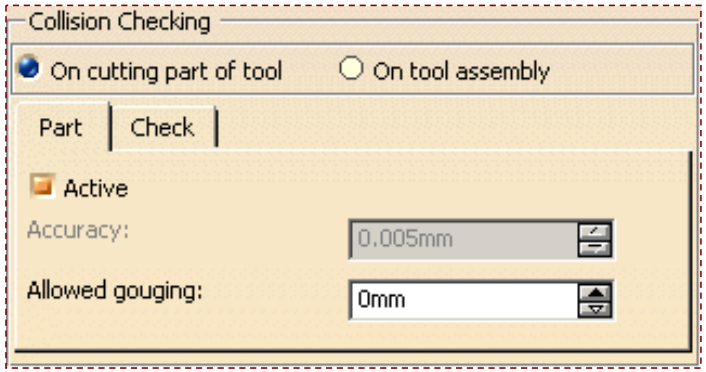
There is no check on contour selection. Bad contour selection may lead to strange tool path.

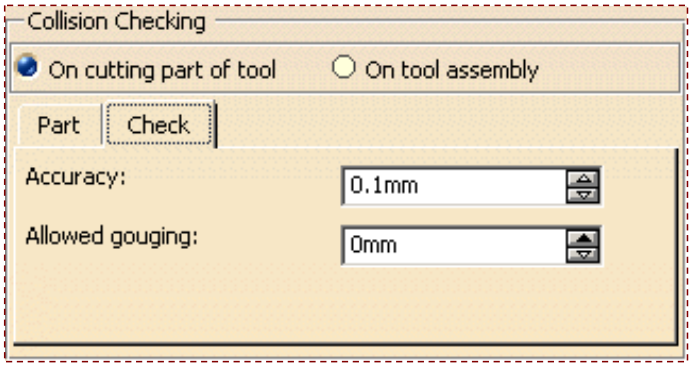
- all contour may lie down on surfaces.
- do not select twice the same element.
- global contour must be closed.

Student Notes:

Collision Checking

- Choose if collision checking is applied on cutting part of the tool (only lc=cutting length) or on the whole assembly (cutting part + shape + holder)
- Activate or not on Part
  - Accuracy is by default initialized with machining tolerance
    - Defines the maximum error to be accepted
  - Allowed gouging need to be set
    - Defines maximum cutter interference
- Set accuracy and allowed gouging on Check tab





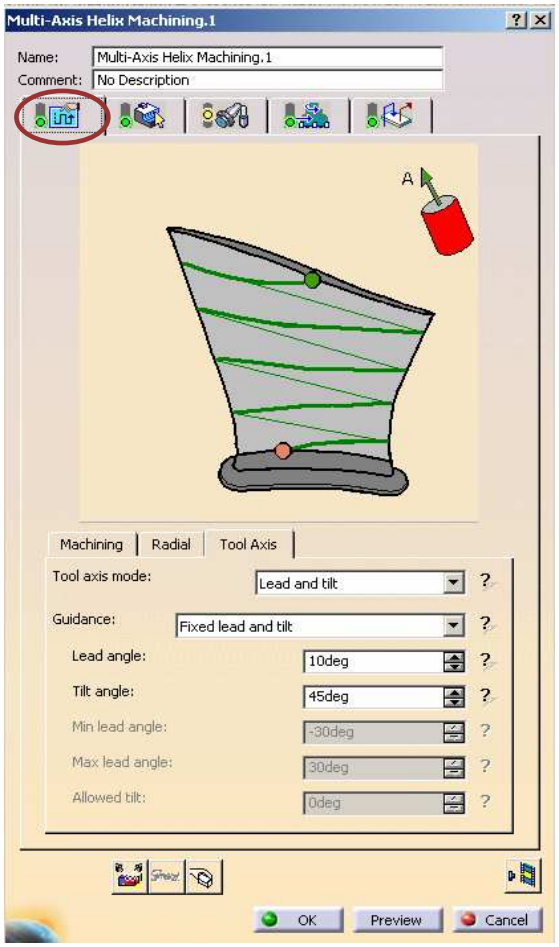
Collision checking may have impact on tool path computation time.

See tool axis variable mode (next foils) to know what is done in case of detected collision.

Student Notes:

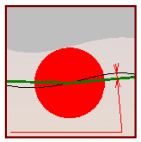
Multi-Axis Helix Machining: Strategy

You will see the Strategy Tab of Multi-Axis Helix Machining.

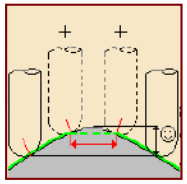


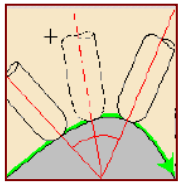
Machining Tab (1/3)

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

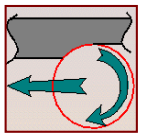


- Max discretization step and angle
  - Maximum distance and angle between two outputted points of tool path (default values are infinite, different settings have to be done according to post-processor and machine feature).

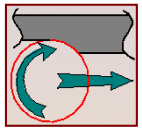


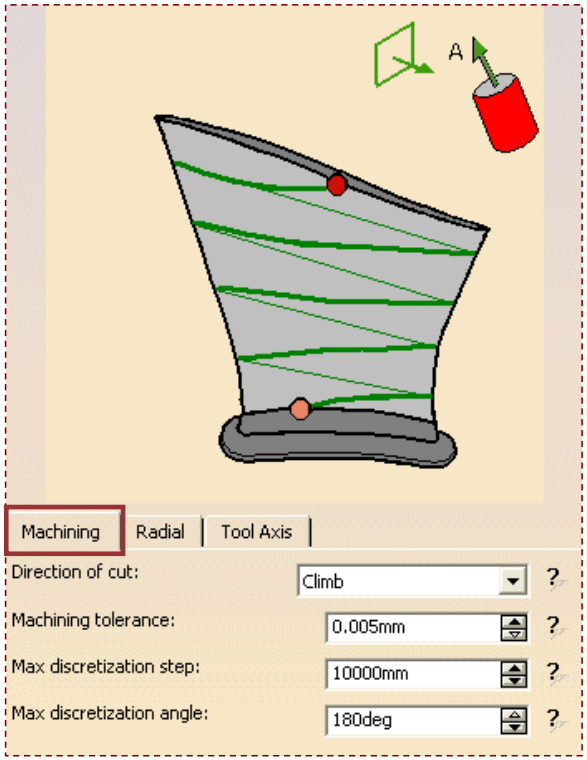


- Direction of cut
  - Climb: The front of the advancing tool cuts into the material first.



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



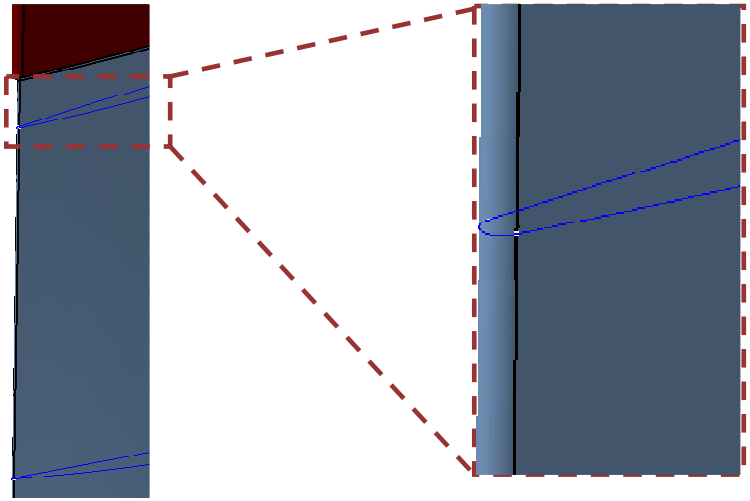


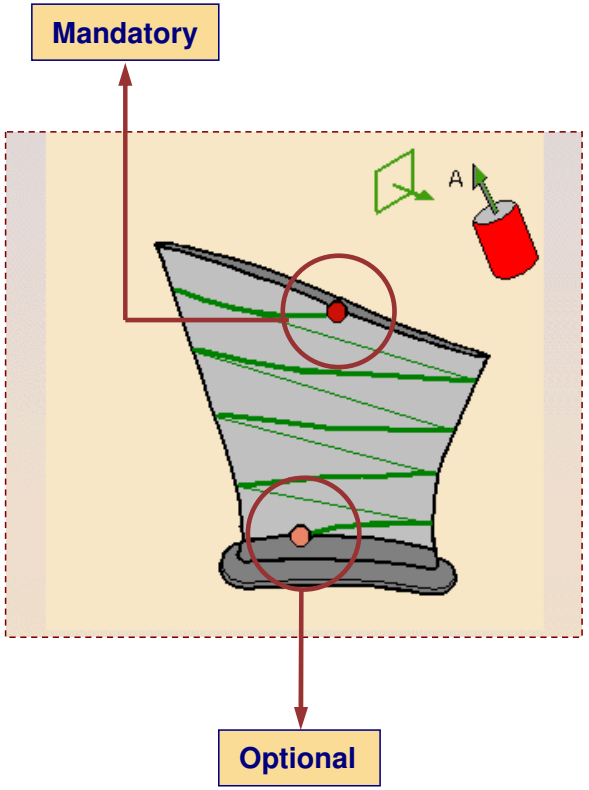
Student Notes:

Student Notes:

Machining Tab (2/3)

- Start and Stop Element
  - One must define a start element (a point) and optionally a stop element (a point)
  - Start point may be pre-defined
    - For example use one extremity of contour near trail or lead edge or
    - Create this point and save it in NC geometry CATPart.



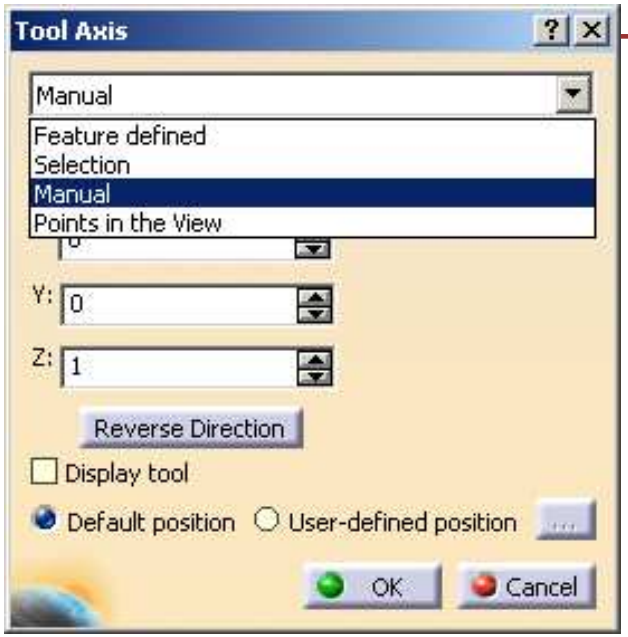


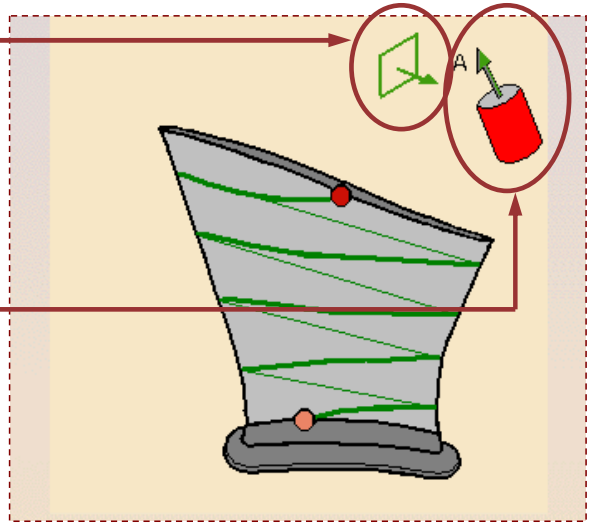
  - Start point may be create on the fly
    - Use MB1 button on part surface to indicate a point. It will be projected on closest contour by algorithm.

Student Notes:

Machining Tab (3/3)

- 4 axis-tilt plane
  - Define machine frozen plane with sensitive plane in picture
- Tool axis
  - Define or modify default tool axis, select sensitive axis in picture

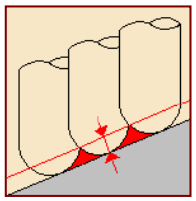


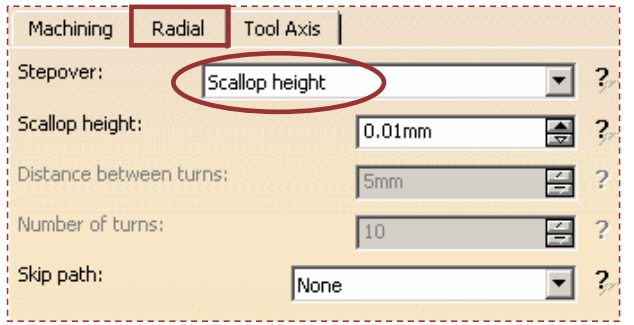


Student Notes:

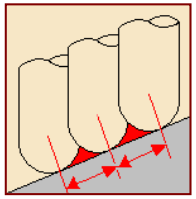
Radial Tab (1/2)

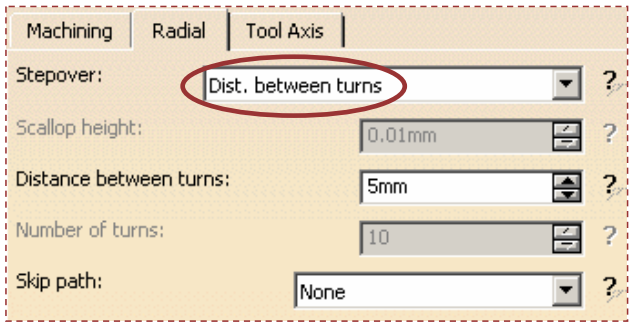
- **Scallop height stepover**
  - Set the maximum scallop height allowed between two paths



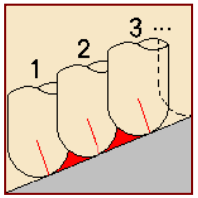


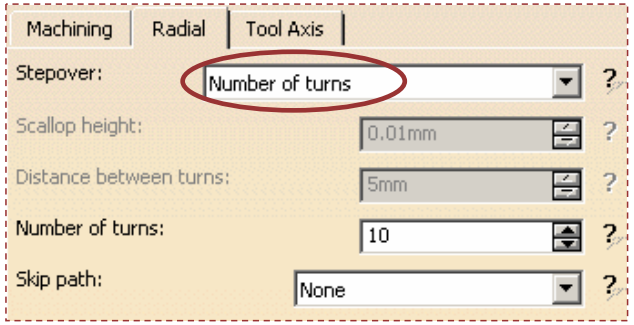
- **Distance between turns stepover**
  - Set the distance between two paths





- **Number of turns stepover**
  - Set a number of turns



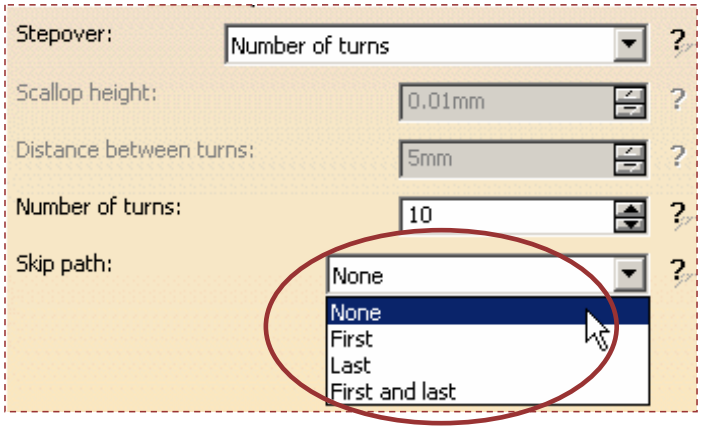


Always start with easy strategy in order to roughly validate your work.
For example start with Nb of turn=10

Student Notes:

Radial Tab (2/2)

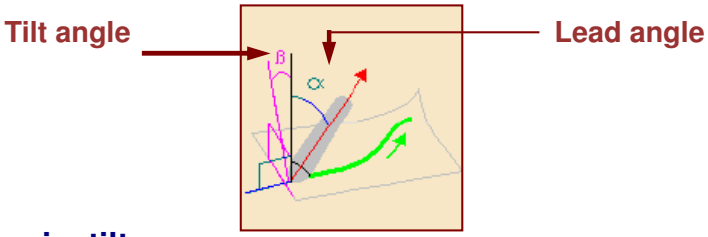
- Skip path
  - In case of machining a part with more than one operation, this option allows to have a better transition between each operation.
  - First, Last or First and last

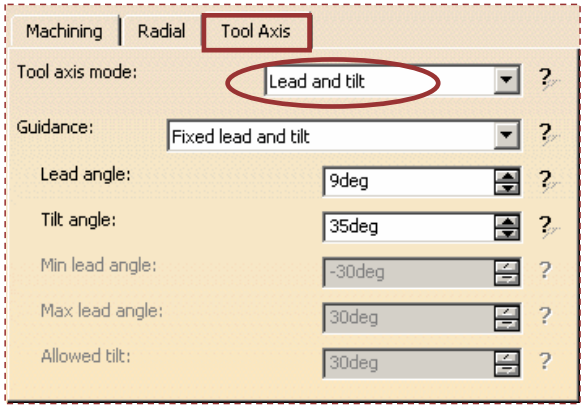


Student Notes:

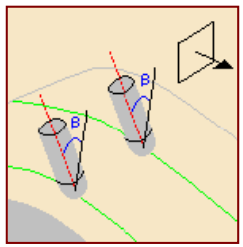
Tool Axis Tab (1/6)

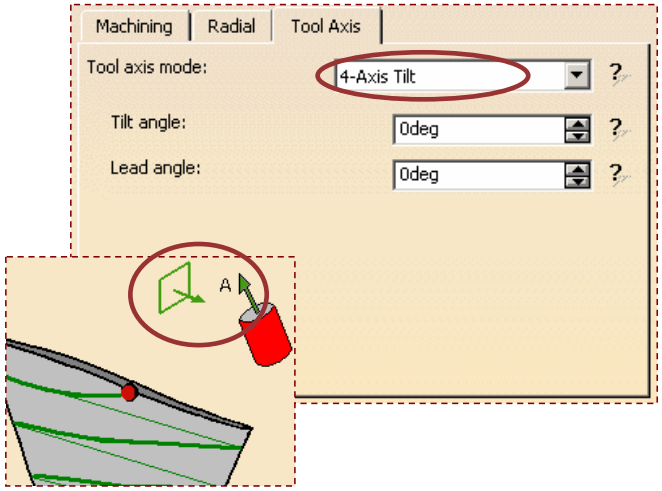
- Lead and tilt
  - For each computed tool path point: local normal at the surface is computed then:
    - Lead angle is set (forward / backward)
    - Tilt angle is set (right/left)





- 4 axis–tilt
  - Define machine frozen plane with sensitive plane in picture
  - According to normal at this plane:
    - Tilt angle is set (right/left)
    - Lead angle is set (forward/backward)





Student Notes:

Tool Axis Tab (2/6)

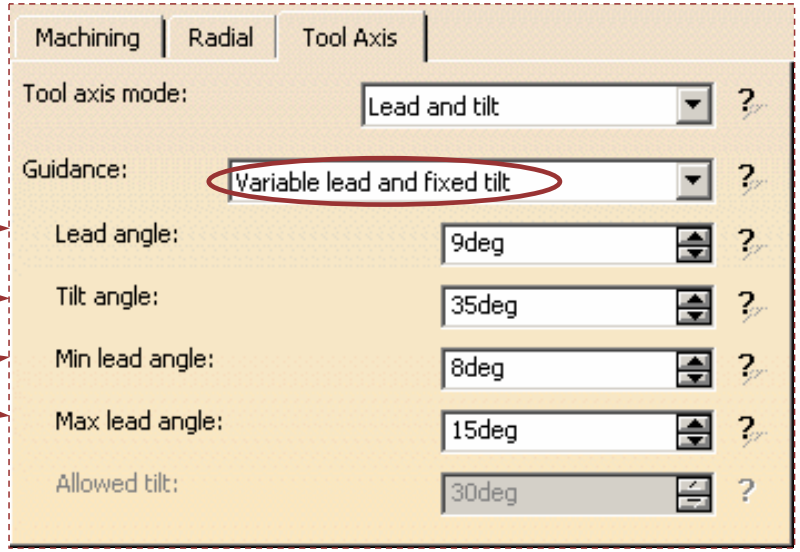
- Lead and tilt variation type
  - Once lead and tilt strategy is chosen, in case of collision checking activated two degraded modes are available:

1. Variable lead and fixed tilt -

Set the reference Lead Angle

Set the fixed Tilt Angle

Set the max and min allowed lead



In this example, reference lead is 9° and could change from 8° to 15°.

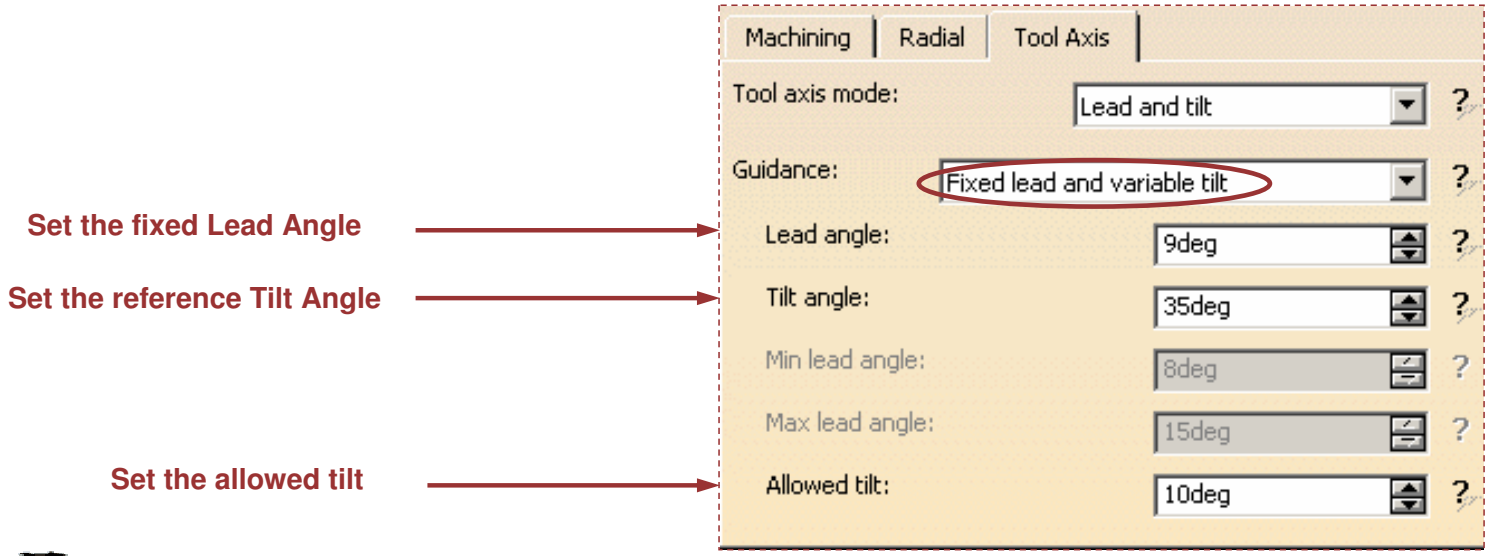
Change in lead would be performed if collision checking is active (part and/or check) and algorithm can compute a better tool position using the allowed change in Lead angle.

Variable lead mode is dedicated to avoid collision between tool rear side and part.

Student Notes:

Tool Axis Tab (3/6)

2. Fixed lead and variable tilt -



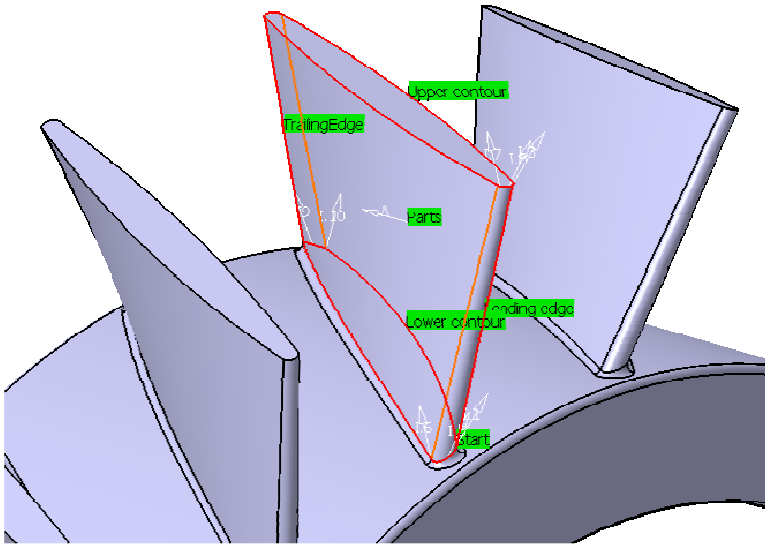
In this example, reference tilt is 35° and could change from 25° to 45°.

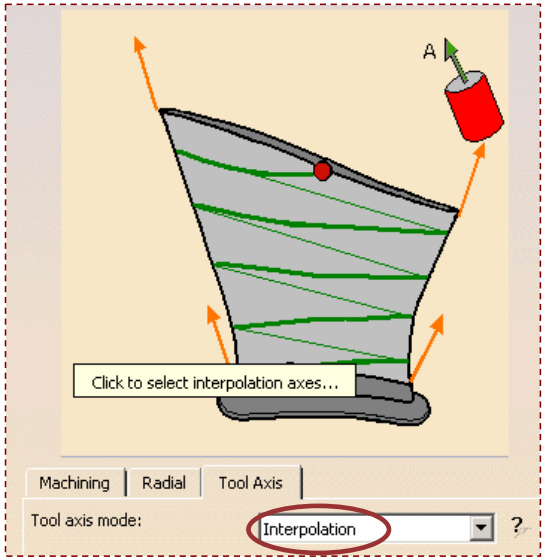
Change in lead would be performed if collision checking is active (part and/or check) and algorithm can compute a better tool position using the allowed change in Tilt angle.

Variable tilt mode is dedicated to ball-end tool machining with risk of collision between tool shape and part.

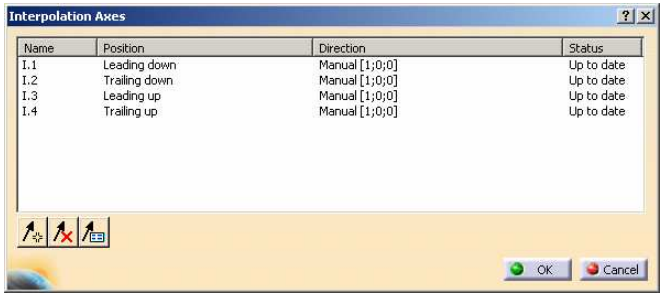
Tool Axis Tab (4/6)

- Interpolation
  - This Tool Axis strategy allows to manually define axes in order to have a better control on tool.
  - This strategy is very useful to avoid collision with others blades.





By default, 4 axes are initialized. You can remove or edit these axes and you can add more axes.



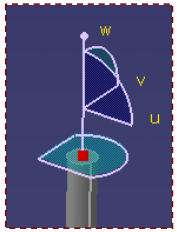
List of default Interpolation axes

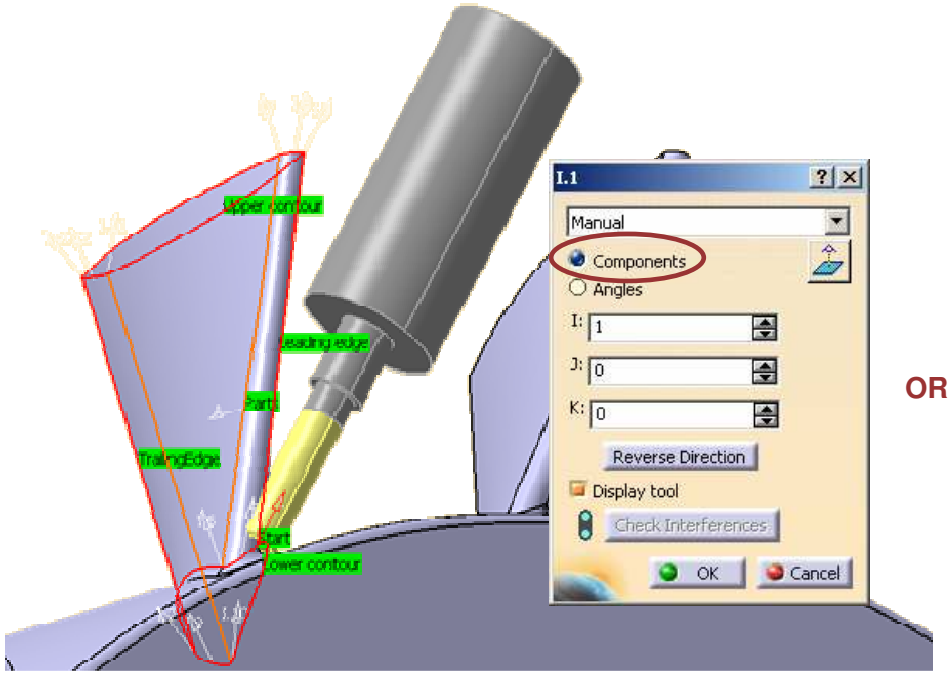
Student Notes:

Student Notes:

Tool Axis Tab (5/6)

- Interpolation: adding / modifying (editing) axes
  - Select existing axis and adjust parameters in dialog box or select existing pre-defined axis (previously created and store in Ncgeometry CATPart).
  - In each case one can use Display tool option to control collision.
  - During axis selection, you can put the compass at the top of the tool (as shown) to adjust roughly the tool axis.

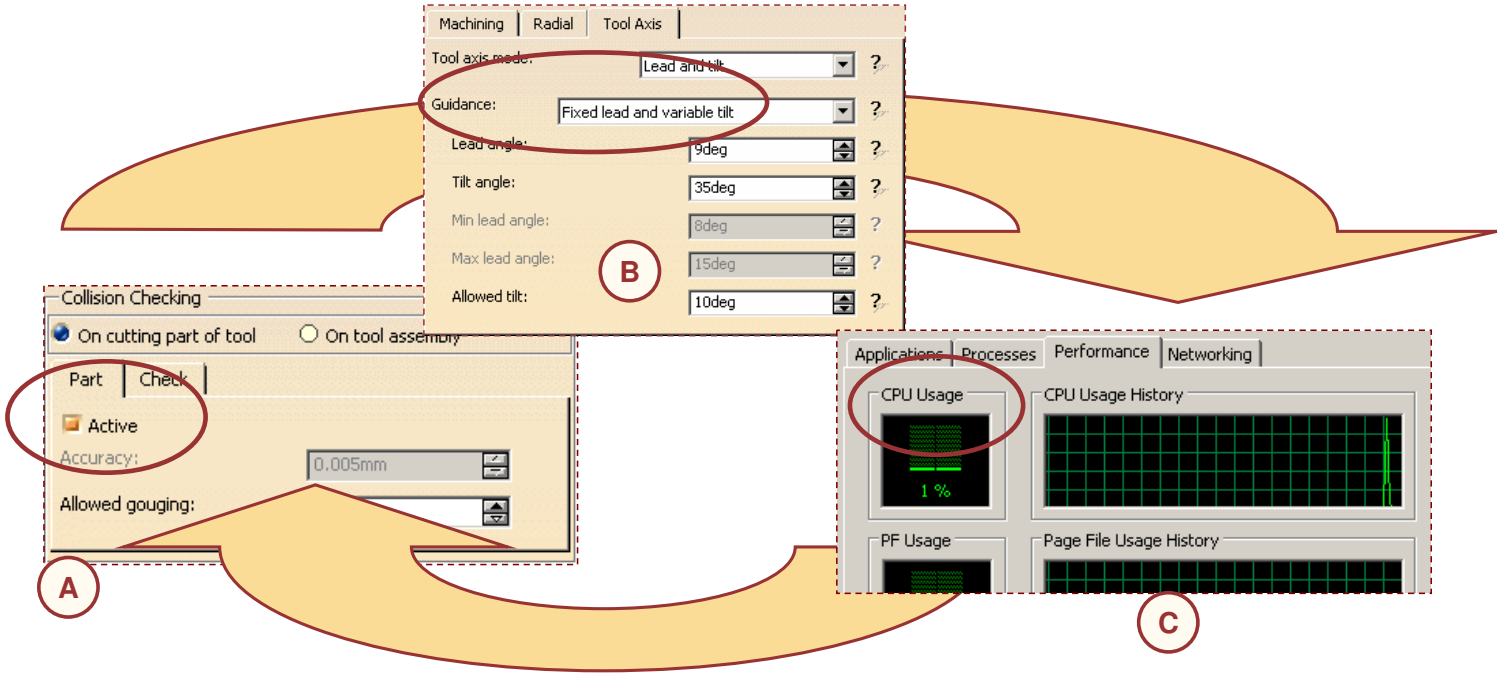




OR

Tool Axis Tab (6/6)

- Lead and tilt variation type
  - Keep in mind following points
    - A Degraded mode (variable tilt or variable lead) are applied only if collision checking is active.
    - B Degraded mode may lead to abrupt tool axis variation and have to be checked with machine capabilities.
    - C Collision checking may have impacts on tool path computation time.



Student Notes:

Student Notes:

To Sum Up

In this course you have seen:

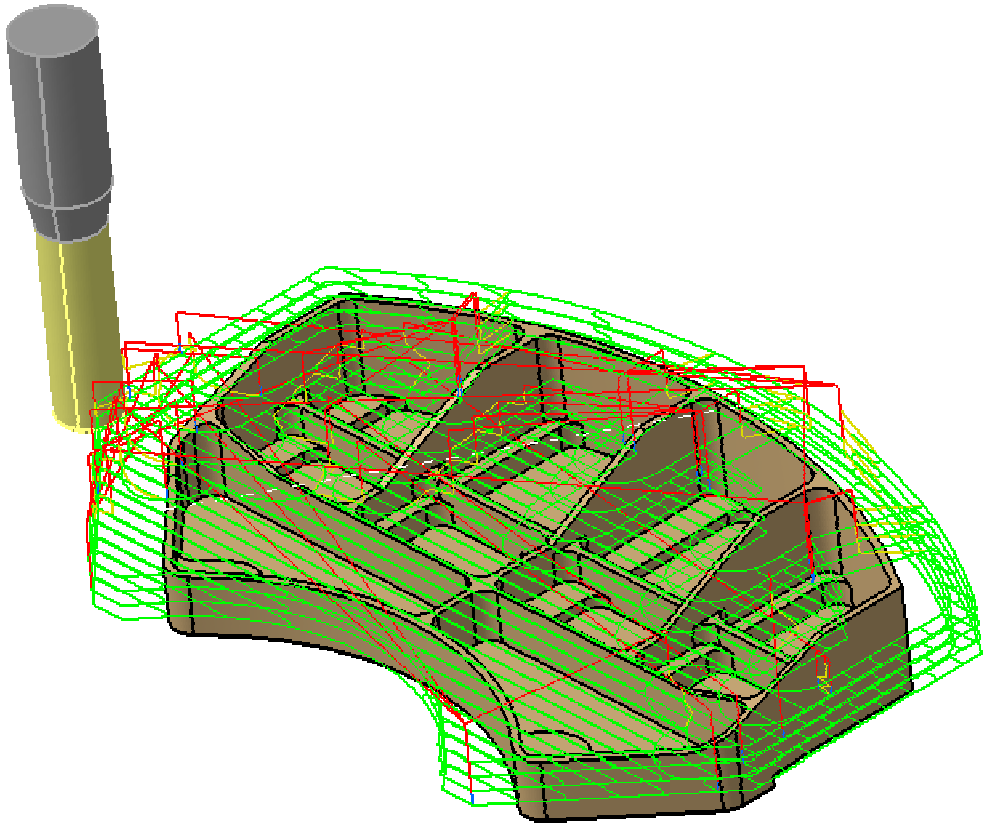
- Necessary geometrical elements to define a Multi-Axis Helix Machining
  - Part
  - Contour
  - Lead and trail edges
  - Start point
- 5 or 4 Axis strategies of MX Helix
  - Lead and tilt (degraded mode available), 4 axis tilt and Interpolation
- Step over management
  - Maximum scallop
  - Distance between path
  - Number of turns

Student Notes:

Cavities Roughing

You will become familiar with the Cavities Roughing principles.

- Introduction
- General Process
- Geometry
- Strategy
- To sum Up



Student Notes:

About Cavities Roughing

Cavities Roughing is mainly used for roughing of Aerospace structural parts.

You can rough machine a part automatically or you can manually select the zones.

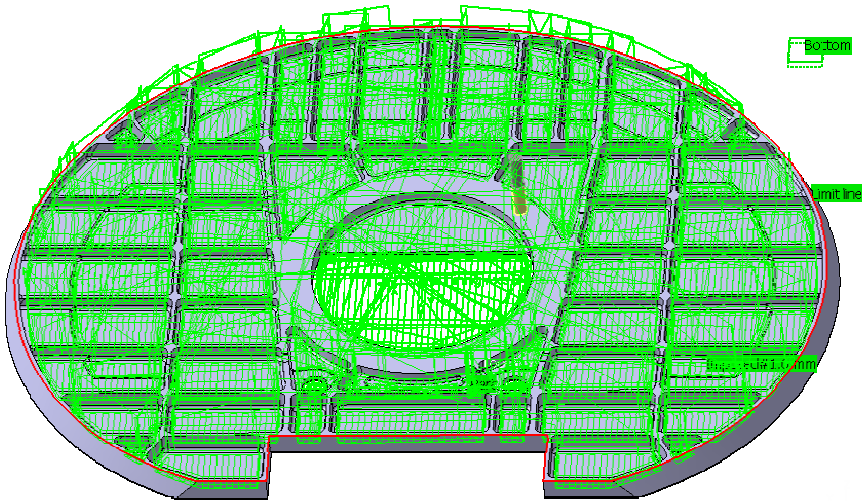
Imposed planes can be inserted to facilitate the forced machining at those levels.

Limiting Contour is useful for roughing and You can use Mask methodology after completion of roughing.

Offset Management can be efficiently done with the available functionalities in Cavities Roughing.

In Cavities Roughing, you can select the area as Outer only, Pockets only or Outer and Pockets. It is possible to leave the thickness on sides and horizontal areas.

Small pockets in the part can be filtered.



Student Notes:

Cavities Roughing: 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 text of comment (optional).

3. Define operation parameters using the 5 tab pages.

Strategy tab

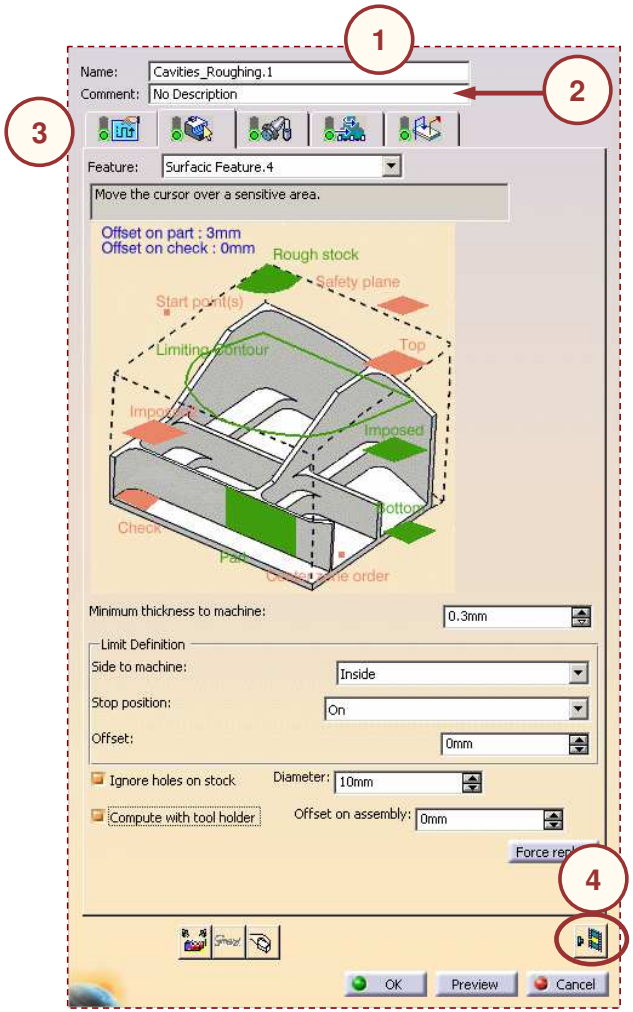
Geometry tab

Tool Definition tab

Feeds & Speeds tab

Transition Paths tab

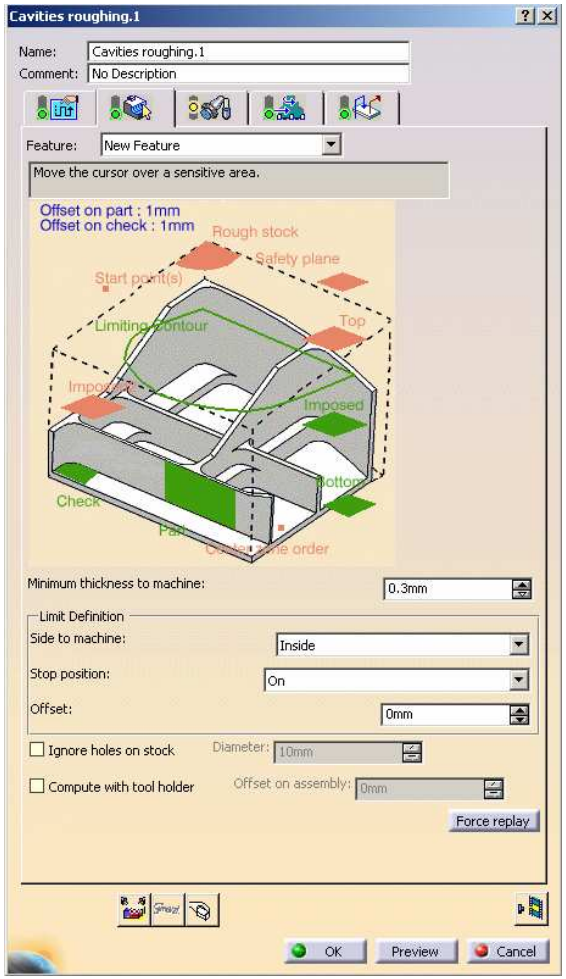
4. Replay and/or Simulate the tool path.



Student Notes:

Cavities Roughing: Geometry

You will become familiar with the options on the Geometry tab of Cavities Roughing.



Student Notes:

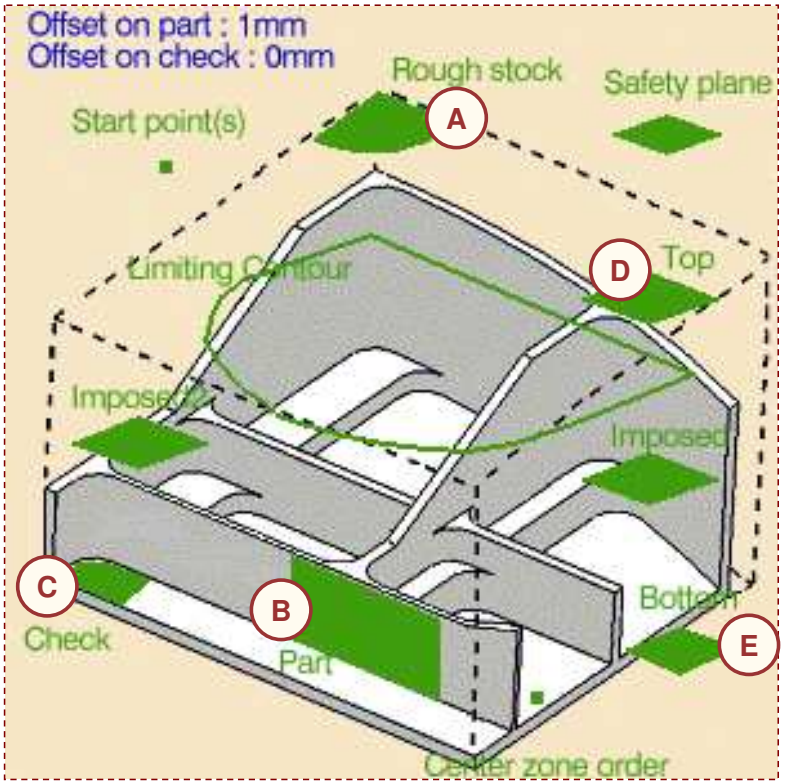
Presentation (1/2)

The Geometry tab includes a sensitive icon dialog box that allows the selection of:

- **A and B : Rough Stock and Part**
  Cavities Roughing operation will remove all stock material in order to obtain final part. Offset can be applied on part.

- **C : Check (optional)**
  Elements to avoid during machining. Offset can be applied on check.

- **D and E : Top and Bottom planes**
  Define them to limit height machining.



Student Notes:

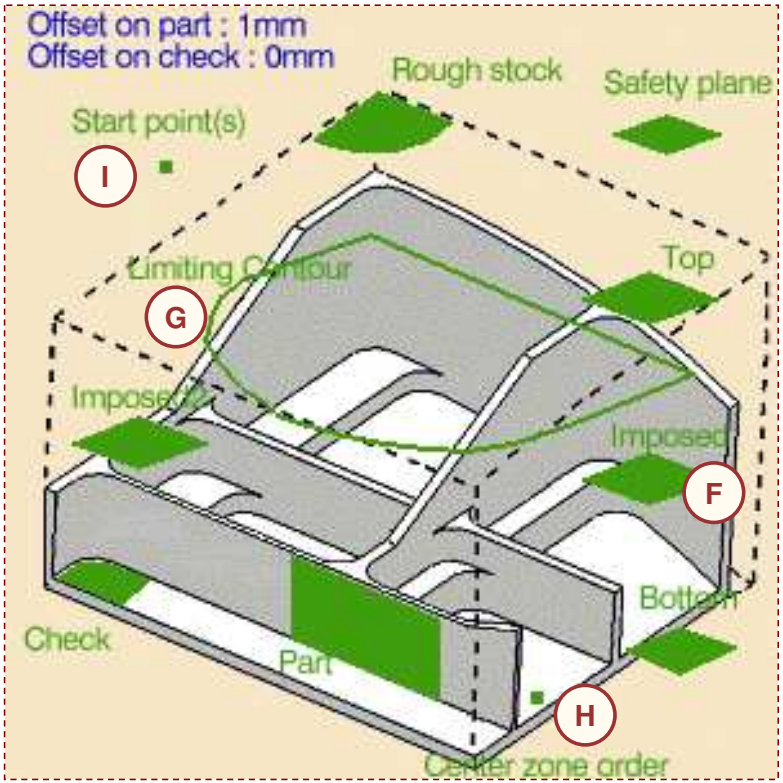
Presentation (2/2)

- F : Imposed planes (two group)
  Force cutter to machine in this plane (global offset can be applied on each group).

- G : Limiting contour
  Re-limit machining area after stock and part Definition.

- H : Center zone order
  Define pocket machining order.

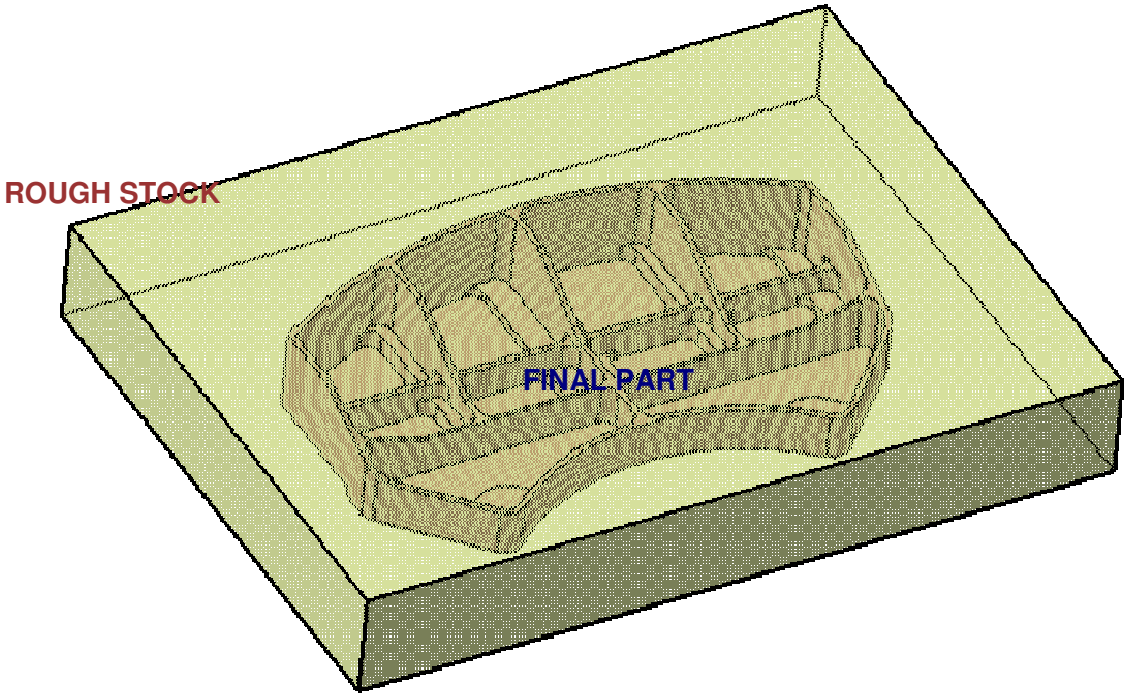
- I : Start Point (optional)
  Impose start point in open area (not in pocket)



Student Notes:

Geometry (1/9): Rough stock and Part

Rough Stock and Part definition example:



Student Notes:

Geometry (2/9): Rework Capability

Rework definition:

Stock definition can be either at Part Operation level or Operation level. To benefit from rework capability, don't define stock at operation level.

Therefore algorithm will compute 'actual stock' taking care all previous operation defined (even non Cavities Roughing operation)

Do not forget to select Force Replay button to update this 'actual stock' if needed.

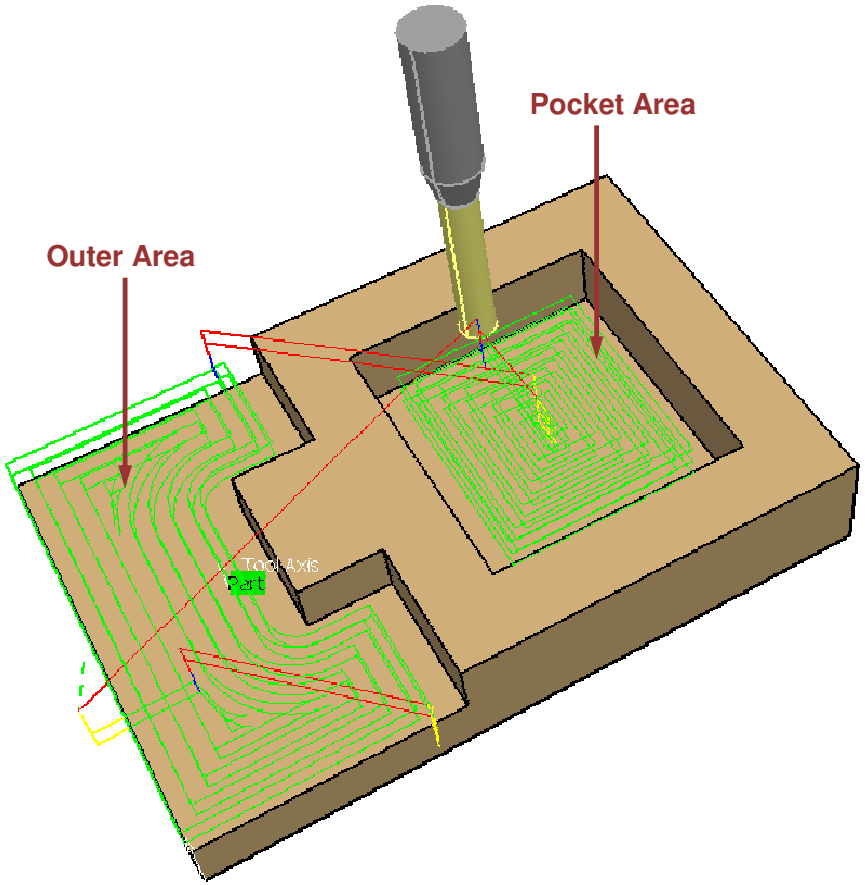
It is recommended to use helical strategy for rework computation in order to have an optimized toolpath.

Minimum thickness to machine parameter:
When using rework capability one can use this parameter that specify the minimum thickness taken into account for computation.

Student Notes:

Geometry (3/9): Outer Part And Pocket Area

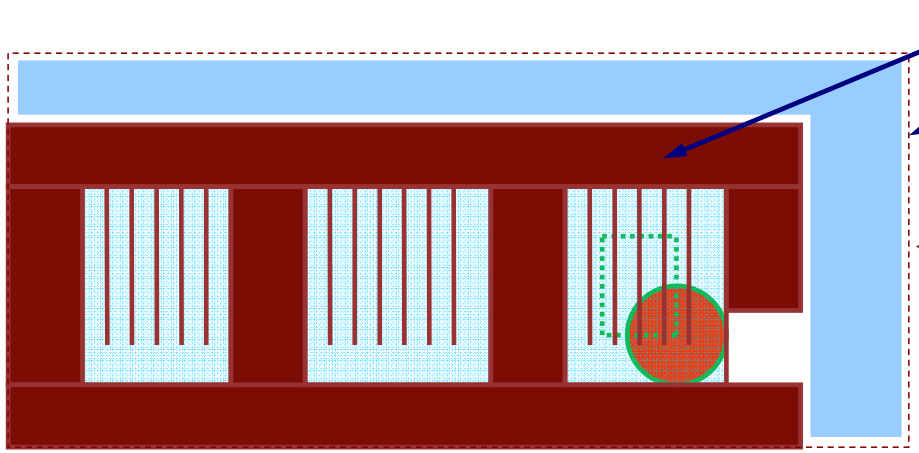
- Outer part and pocket definition:
  - Pocket area: all area tool contouring is touching the part.
  - Outer area: all area which is not pocket area.
- Outer part and pocket notes:
  - It is not only a geometrical concept. It is a function of: Part, Tool diameter and Stock.
  - Be careful, part can be composed of different elements depending of tool diameter. A pocket can become an outer part…see next foil.



Student Notes:

Geometry (4/9): Important Note

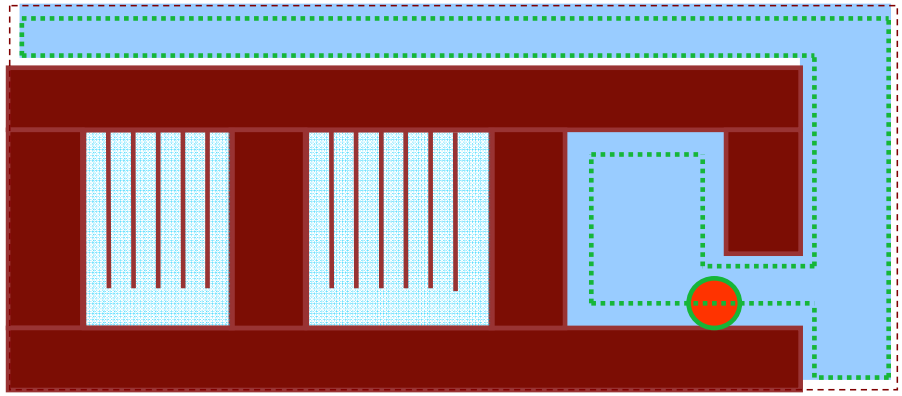
Tool diameter impact on outer and pocket area:



All area tool contouring is touching the part. Hence it is a pocket area.

Outer part area →

Pocket area →

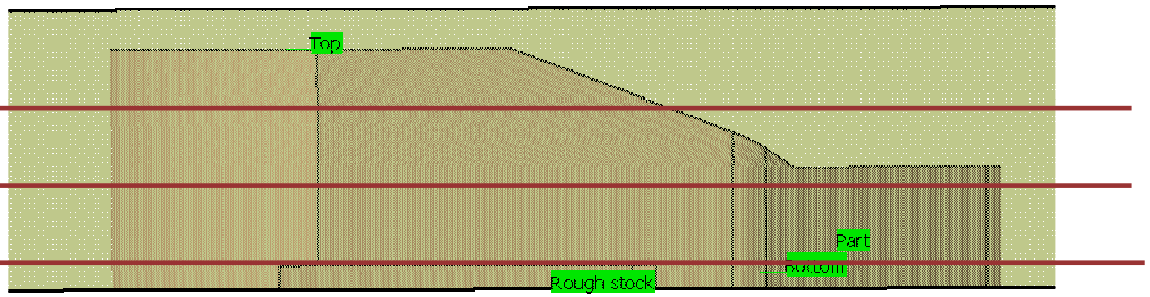


Same geometry but with a smaller tool diameter, this time all area tool contouring is touching the part and the stock, so it's an outer area.

Student Notes:

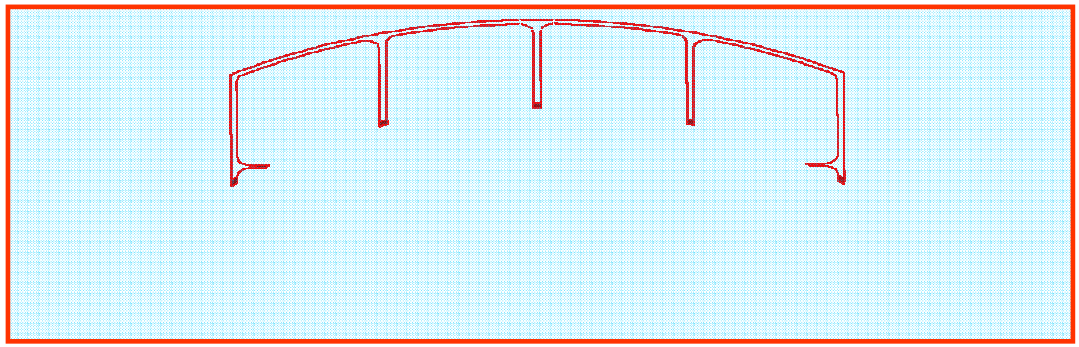
Geometry (5/9): Z Level Plane Impact On Area

Example:



Cut plane 1

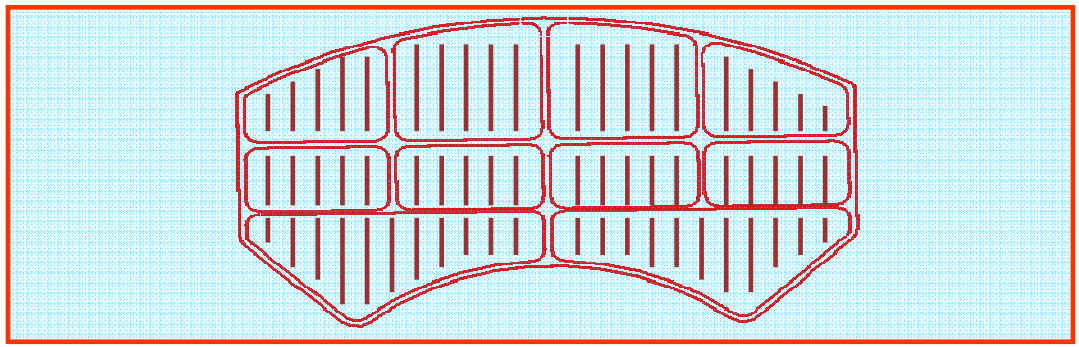
Outer part area



Cut plane 2

Outer part area

Pocket area

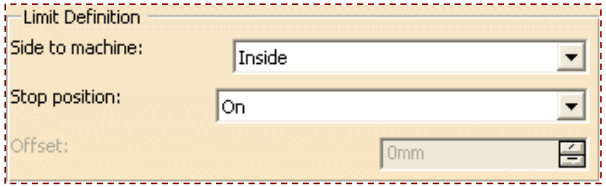


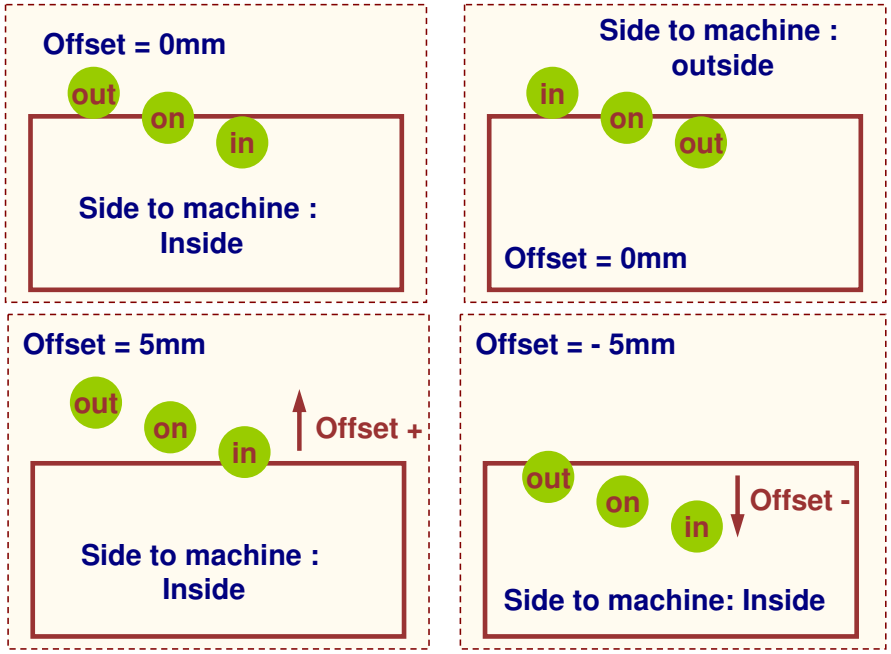
Student Notes:

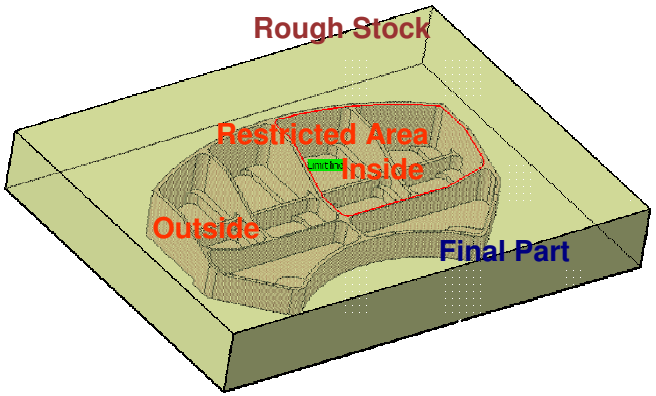
Geometry (6/9): Limiting Contour

Limiting contour is used to restrict machining area to dedicated pockets.
One must define a closed contour with Edge selection wizard then specify Side to machine (inside or outside) and stop position.

Line selection: This wizard allows to select quickly contour elements (navigation).







Positive Offset => offset to outside
Negative Offset => offset to inside

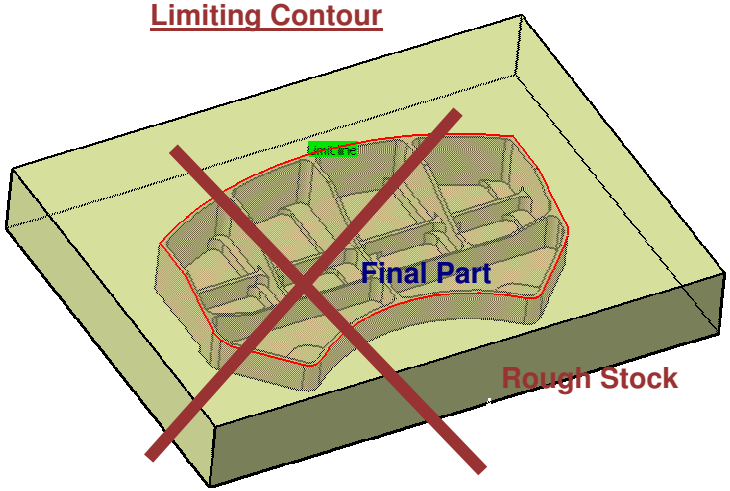
Student Notes:

Geometry (7/9): Mask Methodology

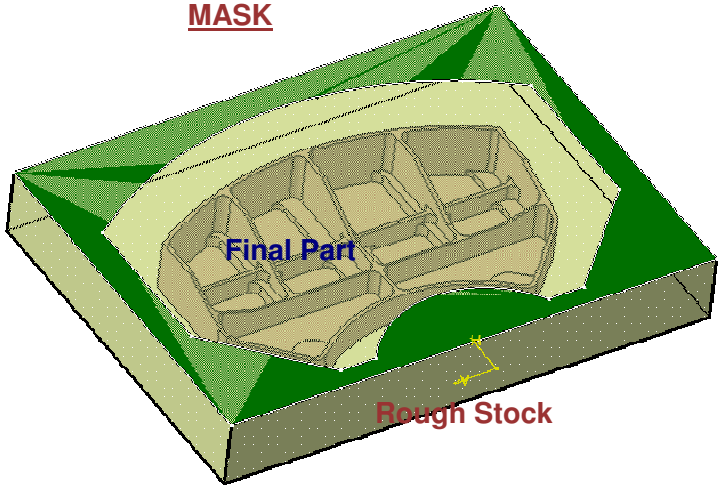
It is not advisable to use limiting contour to describe the part at the end of roughing.
Here the mask methodology is preferred:

Define a mask surface (describing the part at the end of roughing) and select it as part in the user interface.

Limiting Contour



MASK



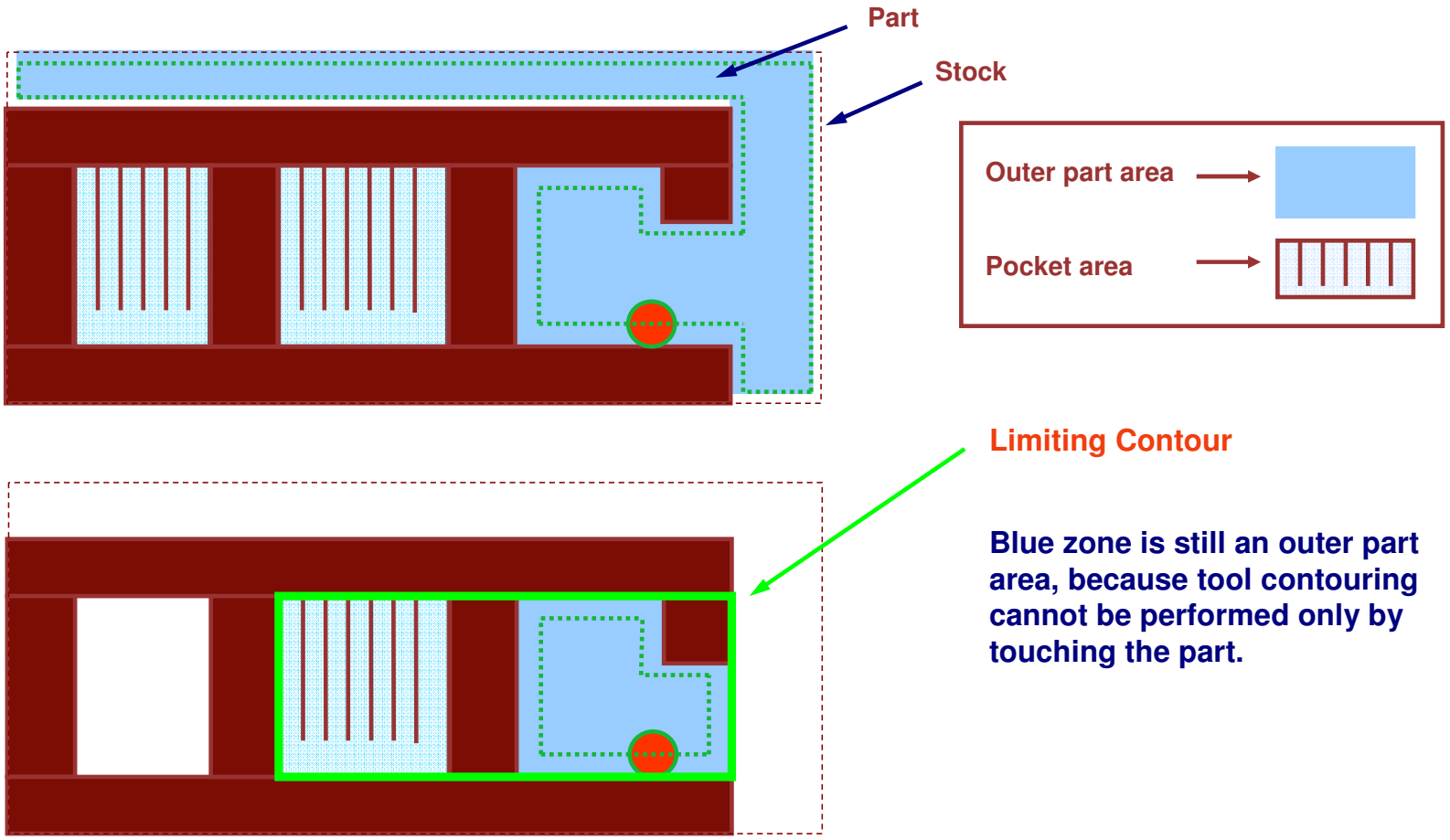
Negative points:
Limiting contour can be crossed by tool tip
Limiting contour impact on outer and part area
Need to manage offset
Offset is function of Tool diameter, thickness on part.

Positive points:
Part elements are not necessarily connected.

Student Notes:

Geometry (8/9): Important Note

Limiting Contour impact on outer and pocket area:

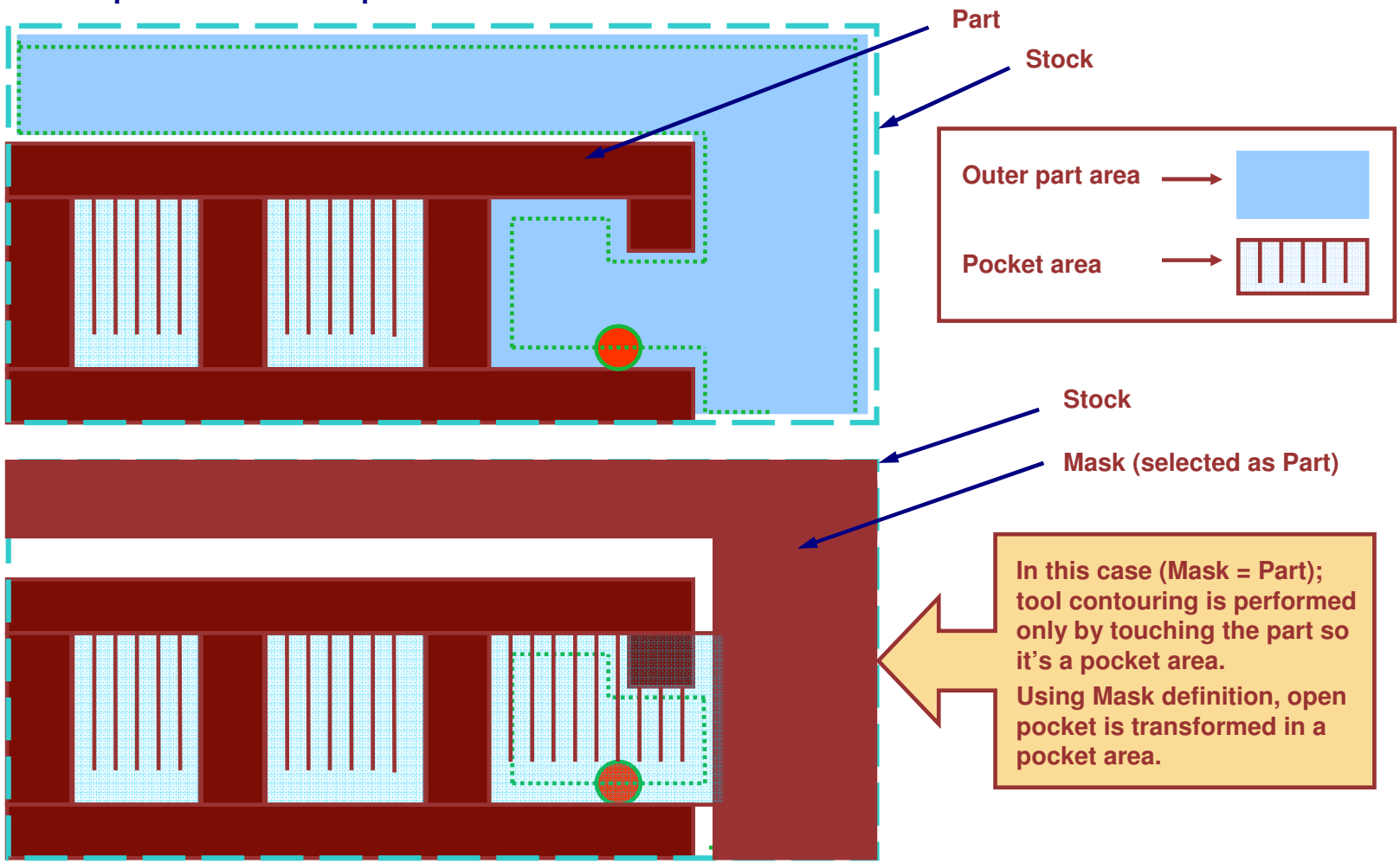


Blue zone is still an outer part area, because tool contouring cannot be performed only by touching the part.

Student Notes:

Geometry (9/9): Important Note

Mask impact on outer and pocket area:

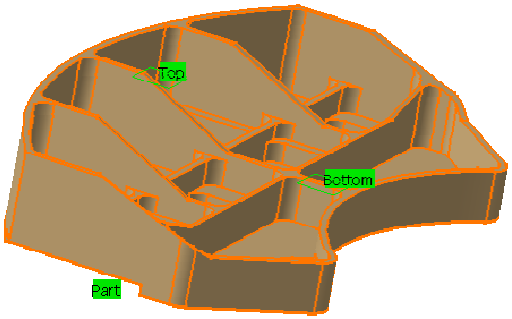


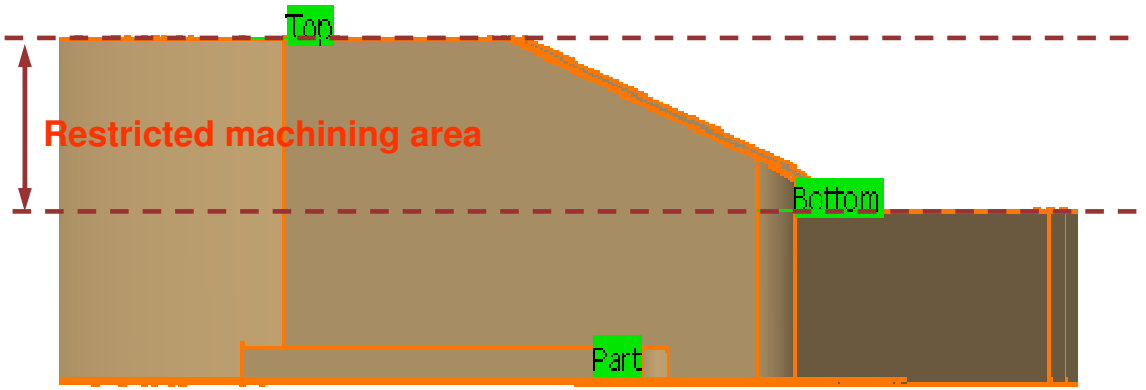
Student Notes:

Planes (1/4): Top and Bottom Planes

Top and Bottom Planes offer capability to restrict height of machining area.

These planes are used in cut depth computation. (see Strategy Tab chapter)



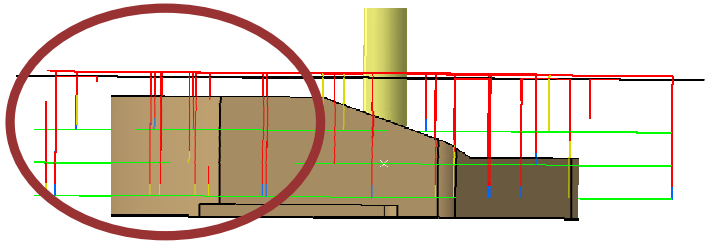


Student Notes:

Planes (2/4): Imposed Planes

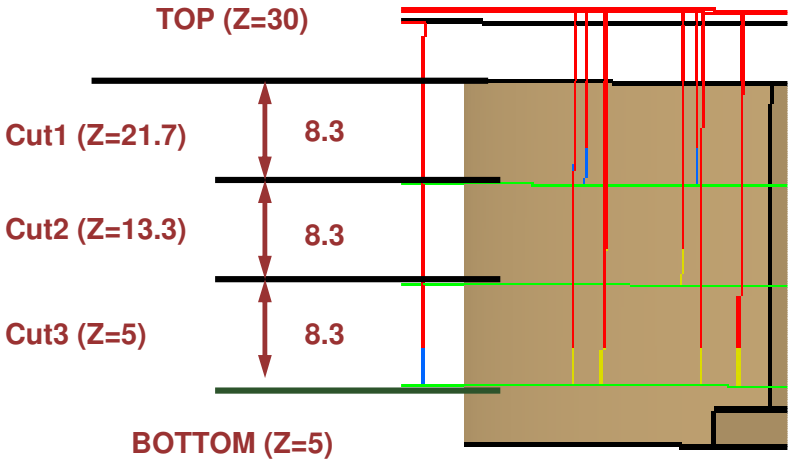
Top and bottom planes with maximum depth of cut allow to define cutting planes.

Adding to them, it is possible to define Imposed cutting planes, manually or using auto search on part. Imposed planes are the planes to which the cutter must positively reach.



Initial step: top and bottom planes selected, max. depth of cut = 10

=> 3 Cut plane automatic computation

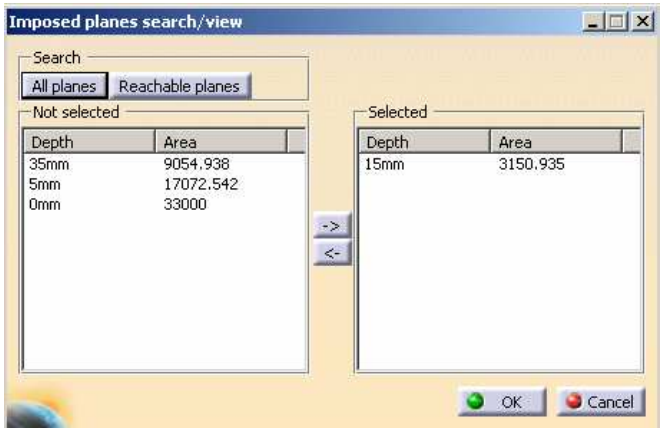


Student Notes:

Planes (3/4): Imposed Planes

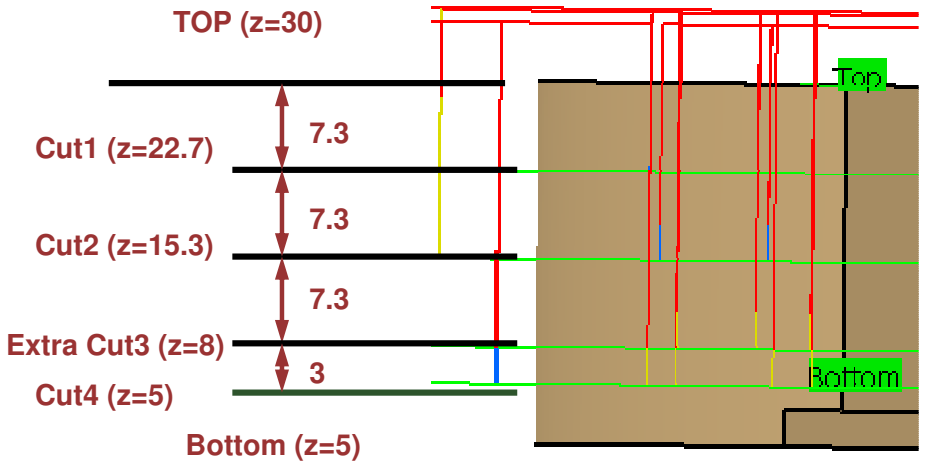
Adding imposed plane with search plane capability:

Select right mouse button on imposed plane sensitive picture then select Search/View menu, the window as shown below will be displayed:



Z= 8 imposed plane added

=> Cut plane 1 and 2 re-computed, extra cut (z=8) added.



Student Notes:

Planes (4/4): Notes

Offset:

All planes (top, bottom, imposed) can be modified using offset capability.
Cutting plane will always strictly respect the offset plane.
Two groups of imposed planes are existing in sensitive picture thus allowing to define two different offsets on imposed planes.

Adding Imposed Plane with Search/View capability:

Scanning is performed on all planar surfaces of the part or only the planes that can be reached by the tool you are using (small pockets and counter-draft area are skipped)
Be careful, offset on imposed planes has to be greater than the global offset on part, otherwise it will not be respected.

Adding imposed plane manually:

Any plane can be selected (physical part plane, plane created in WFS workbench etc)

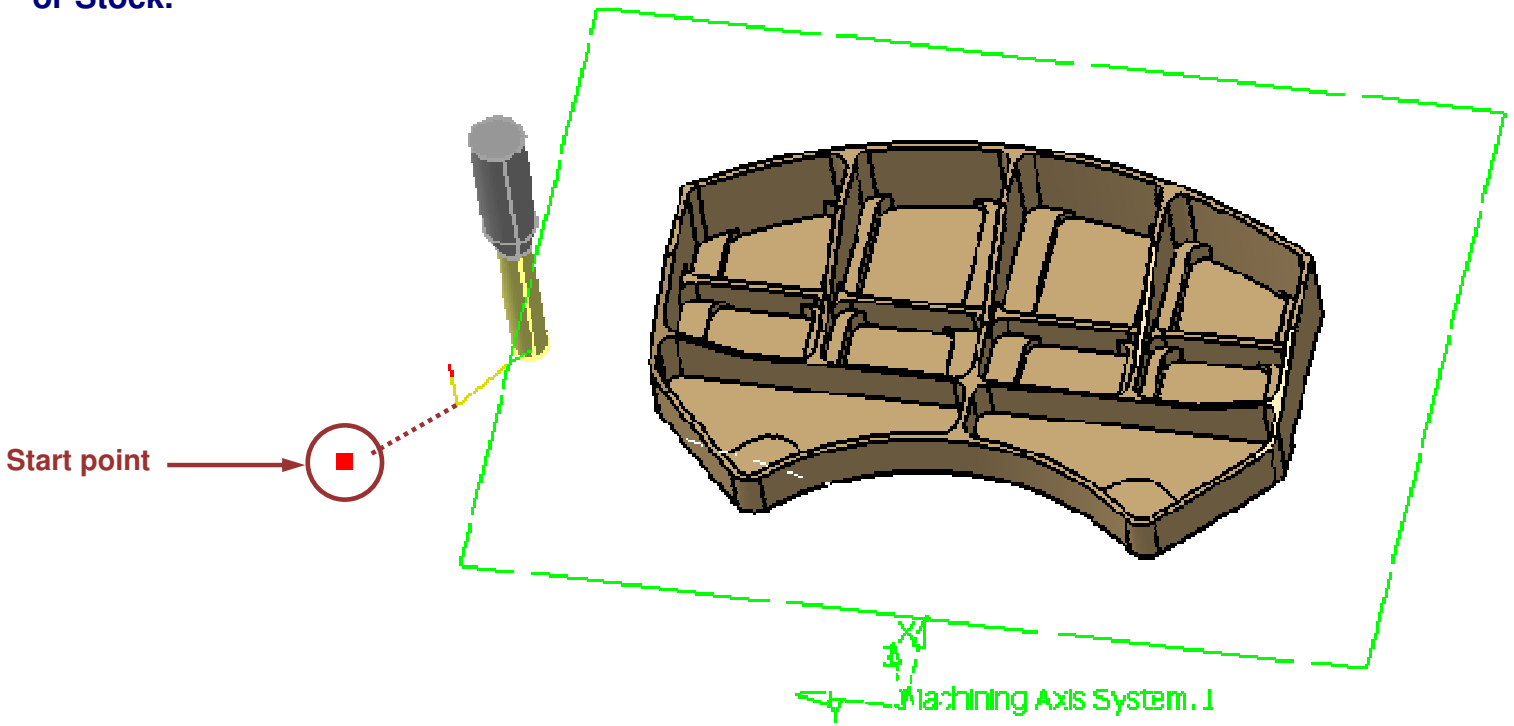
Selection:

System automatically check if selected plane is normal with tool axis (e.g. if plane selection is refused, check operation tool axis)

Student Notes:

Start Point And Zone Order

- Start point restrictions:
  - Only for outer part area (no pocket).
  - Only helical mode.
  - Defined point must not be in collision with Part or Stock.



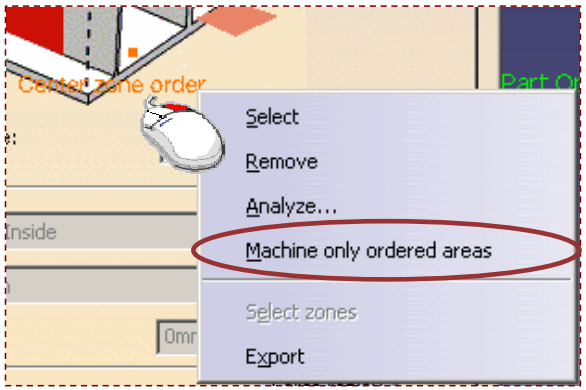
Student Notes:

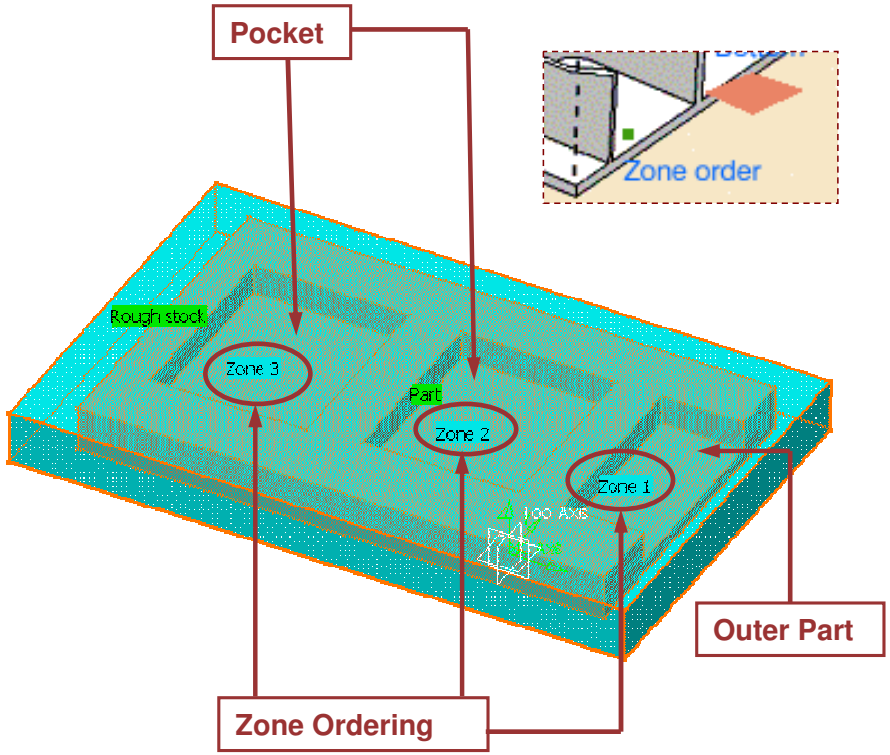
Zone Order

Zone order definition :

It is a capability to define pocket order machining (either outer part or pocket).
It is used to manage stress on part for example.

Zones will be machined in the selected order.
It is possible to machine only selected zones.
(MB3 on Zone Order)

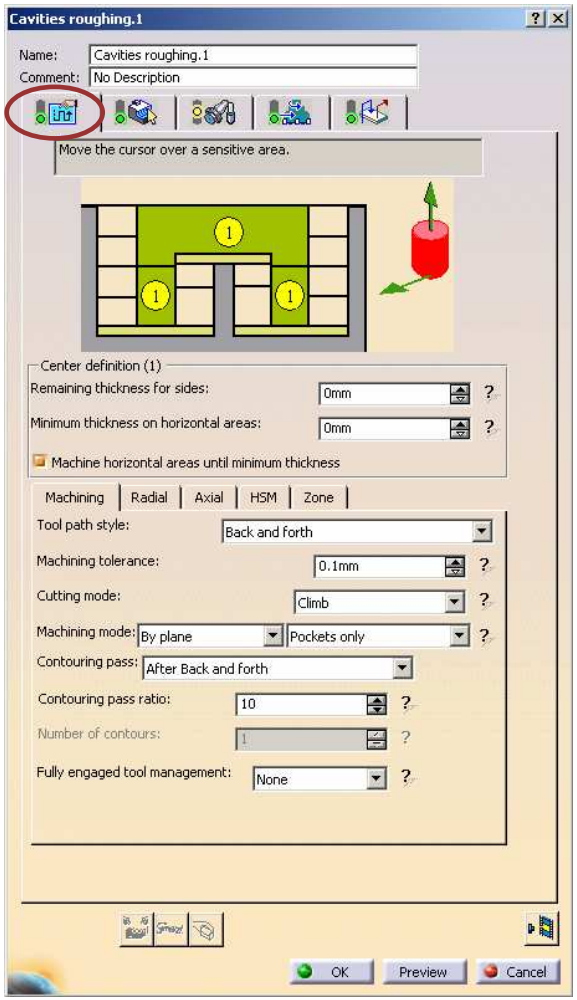




Student Notes:

Cavities Roughing: Strategy

You will learn the options in the Strategy Tab of Cavities Roughing.

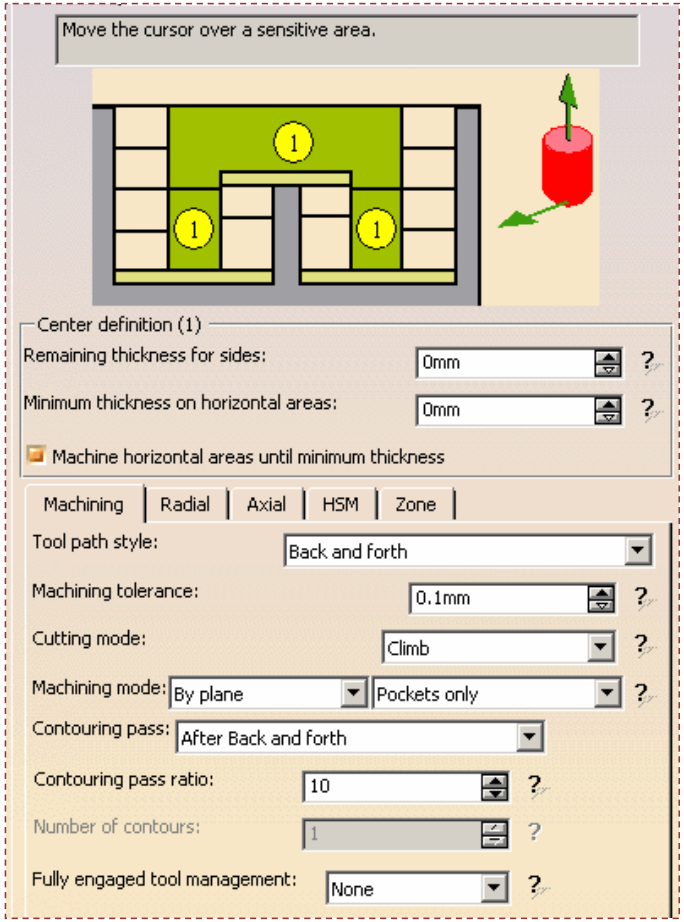


Student Notes:

Presentation

This Tab Page allows to define:

- **Thicknesses on sides and horizontal area.**
- **Offset Management in detail.**
- **Machining, Radial, Axial, HSM and Zone tabs.**
- **Tool axis and cutting directions (sensitive picture).**

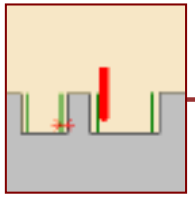


Student Notes:

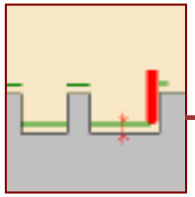
Center Definition (1/3)

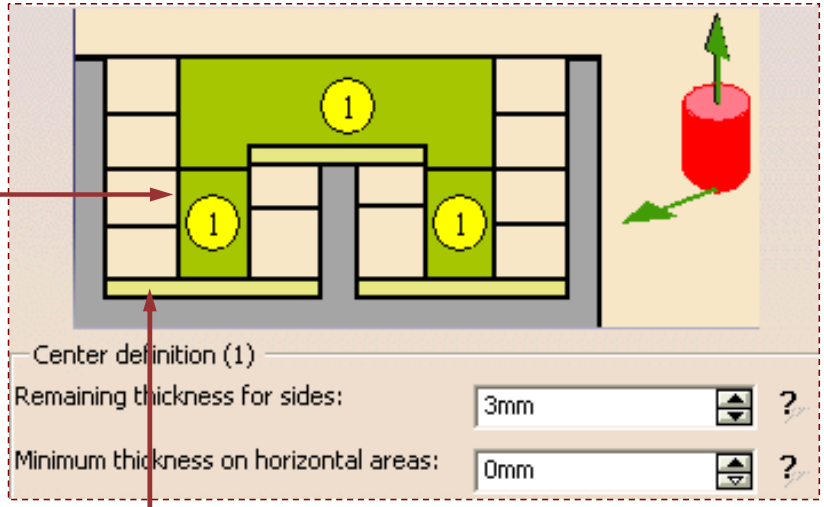
'Center' is roughing the part by leaving thicknesses on sides and horizontal areas.

One can define remaining thickness on sides.

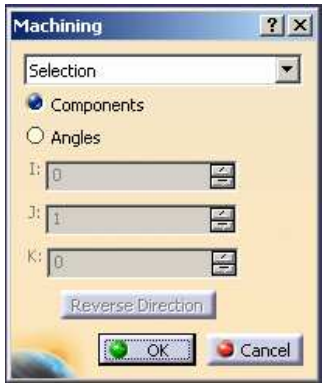


And minimum thickness on horizontal area

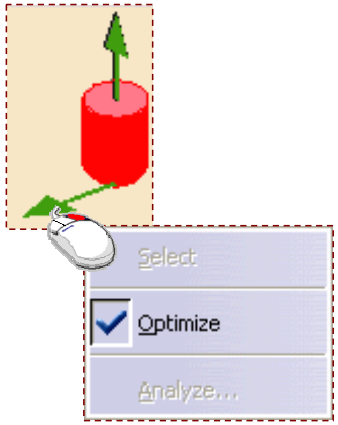




In Back and Forth strategy, machining direction can be set manually using axis definition dialog box. It can be set automatically using optimize option (right mouse button menu).



OR

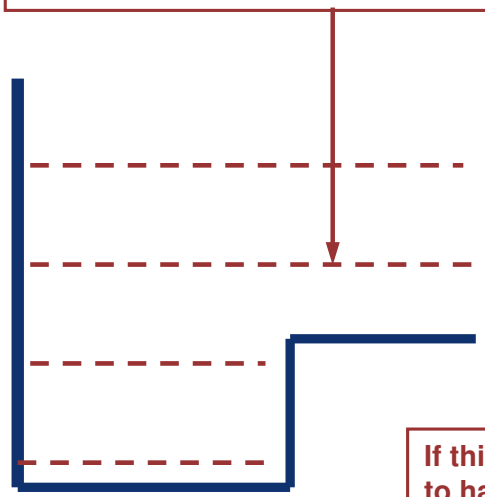


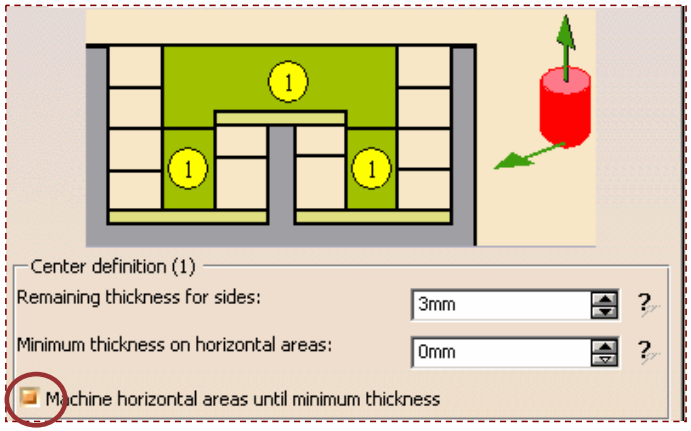
Student Notes:

Center Definition (2/3)

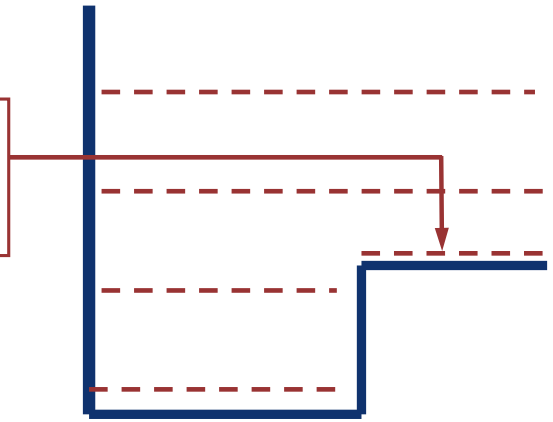
Machine horizontal areas until minimum thickness option:

Depending on cutting plane computed, horizontal area may have till one cut depth remaining material. This cut depth can be machined by using 'Machine horizontal areas until minimum thickness.'





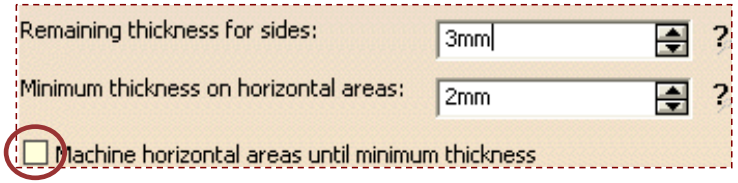
If this option is activated, it will force to have one extra path on this horizontal area to respect minimum thickness.

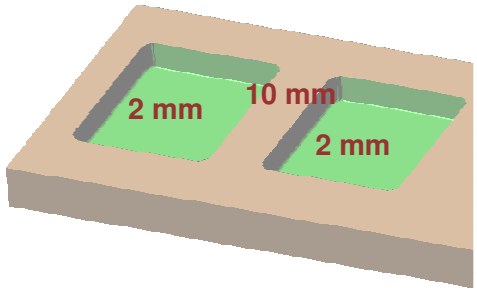


Student Notes:

Center Definition (3/3)

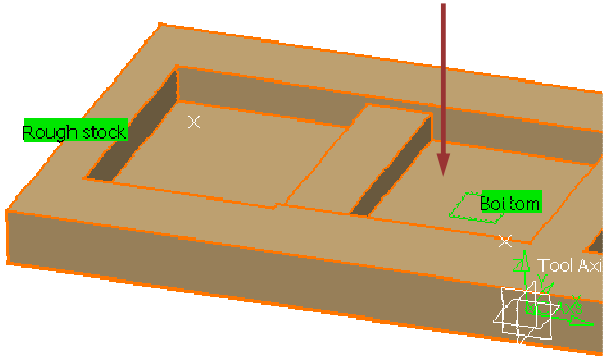
Machine horizontal areas until minimum thickness example:

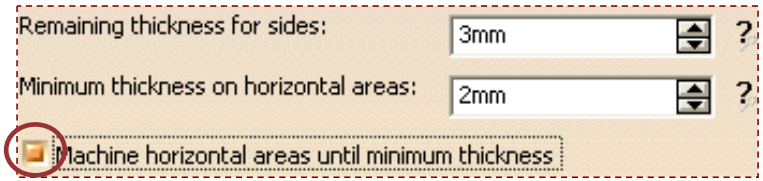


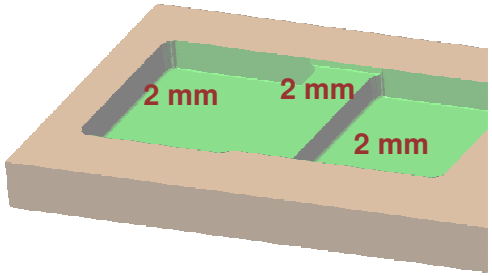


Machine horizontal areas until minimum thickness - Not activated

Bottom plane = bottom of pocket + 2mm offset







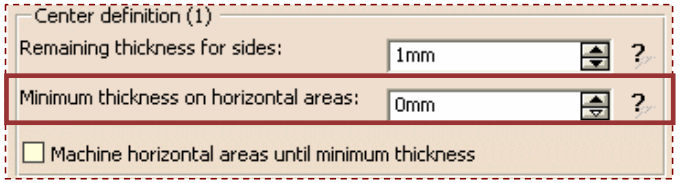
Machine horizontal areas until minimum thickness - Activated

Student Notes:

Offset Management: Case 1- Part Offset

Parameters:

- Part offset =1mm (blue) → forbidden to go under this value
- Max depth of cut = 3mm



Condition to be respected:
Offset on each horizontal area ≥ part offset

Compute of the remaining material depth on horizontal areas =
H-D*N ≥ Part offset + Min thickness on horizontal areas

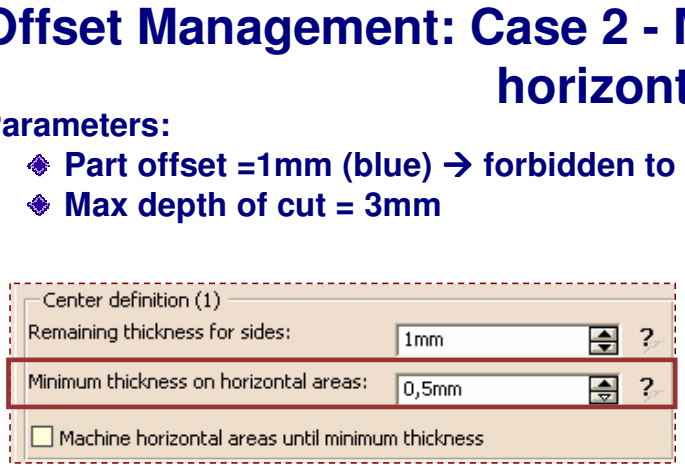
H : depth to remove
D : max depth of cut
N : number of level

Student Notes:

Offset Management: Case 2 - Minimum thickness on horizontal areas

Parameters:

- Part offset =1mm (blue) → forbidden to go under this value
- Max depth of cut = 3mm



Condition to be respected:
Offset on each horizontal area ≥ part offset + Min thickness on horizontal areas (1.5mm)

Compute of the remaining material depth on horizontal areas =
H-D*N ≥ Part offset + Min thickness on horizontal areas

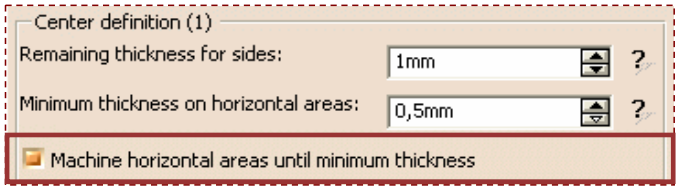
H : depth to remove
D : max depth of cut
N : number of level

Student Notes:

Offset Management: Case 3 - Machine horizontal areas until minimum thickness

Parameters:

- Part offset =1mm (blue) → forbidden to go under this value
- Max depth of cut = 3mm



Condition to be respected:
Offset on each horizontal area = part offset + Min thickness on horizontal areas (1.5mm)

Compute of the remaining material depth on horizontal areas = Part offset + Min thickness on horizontal areas

Added plane to reach 1.5 mm On each horizontal area

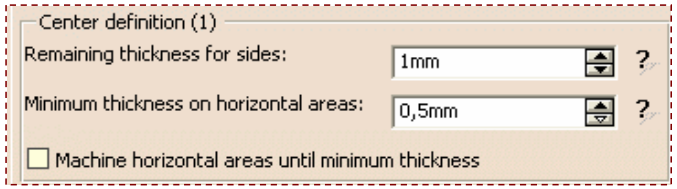
H : depth to remove
D : max depth of cut
N : number of level

Student Notes:

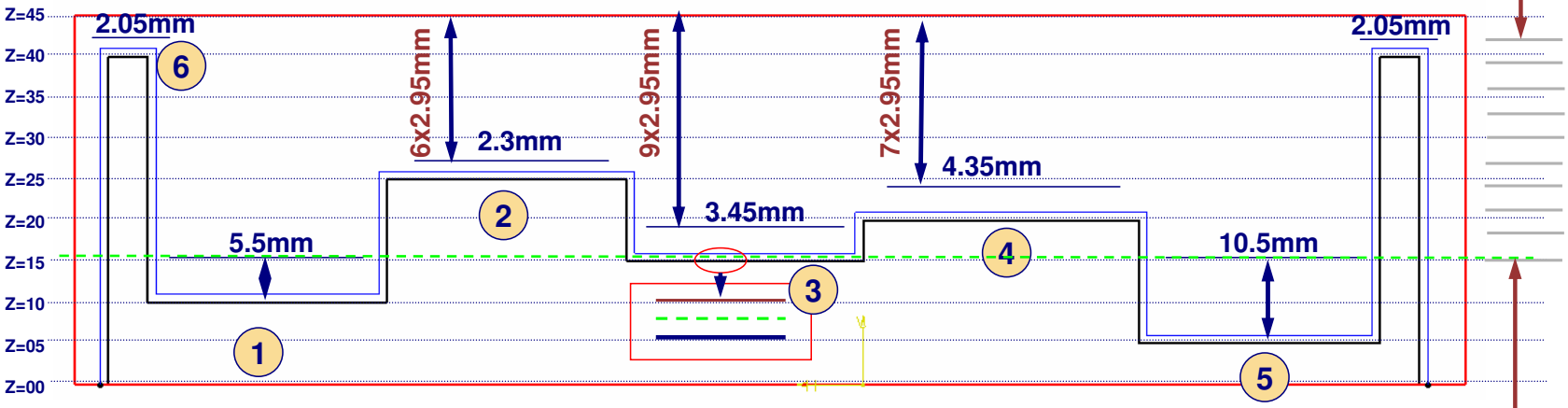
Offset Management: Case 4 - Bottom Plane

Parameters:

- Part offset =1mm (blue) → forbidden to go under this value
- Max depth of cut = 3mm
- Define bottom plane with 0.5mm offset -------- (Z=15.5)



Condition to be respected:
Offset on each horizontal area ≥ part offset + Min thickness on horizontal areas (1.5mm)



1. Recomputed depth to have regular depth of cut: H(top-bottom)/N closest than max depth of cut = 2.95 mm
2. Compute of the remaining material depth on horizontal areas ≥ part offset + Min thickness on horizontal areas

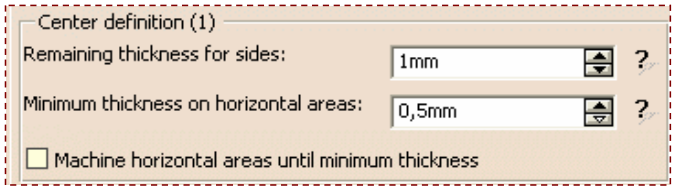
H (top-bottom): depth to remove from top of the stock to bottom plane
N: number of level
The bottom path is done only in zones 1 & 5.

Student Notes:

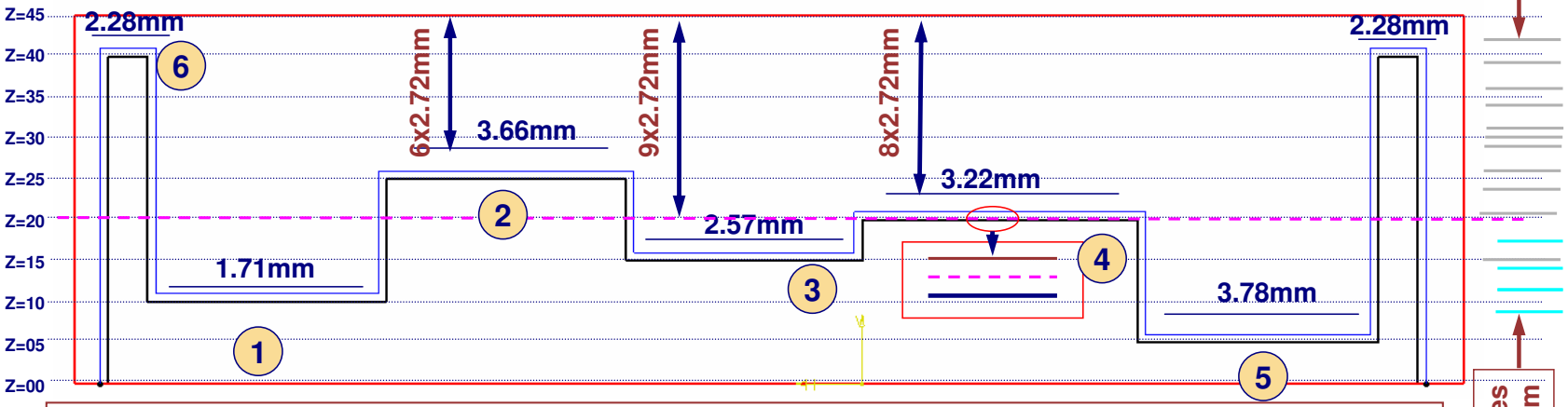
Offset Management: Case 5 - Imposed Plane

Parameters:

- Part offset =1mm (blue) → forbidden to go under this value
- Max depth of cut = 3mm
- Define Imposed plane with 0.5mm offset - - - - - - - (Z=20.5)



Condition to be respected:
Offset on each horizontal area ≥ part offset + Min thickness on horizontal areas (1.5mm)



Computed planes Each 2.72 mm

Computed planes Each 2.93mm

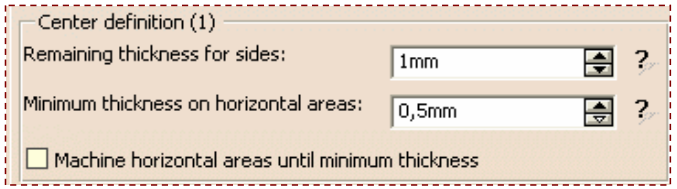
1. Recomputed depth to have regular depth of cut between imposed planes
until imposed plane: Recompute depth: H (top-imposed plane)/N closest than max depth of cut = 2.72 mm
after imposed plane: Recompute depth: H (imposed plane-last plane)/N closest than max depth of cut = 2.93 mm
2. Compute of the remaining material depth on horizontal areas ≥ part offset + Min thickness on horizontal areas
N : number of level
The imposed plane path is done only in zones 1, 3 & 5.

Student Notes:

Offset Management: Case 6 - Top Plane

Parameters:

- Part offset =1mm (blue) → forbidden to go under this value
- Max depth of cut = 3mm
- Define Imposed plane with 1mm offset - - - - - - - (Z=35)



Condition to be respected:
Offset on each horizontal area ≥ part offset + Min thickness on horizontal areas (1.5mm)

1. Recomputed depth to have regular depth of cut between top and bottom planes (here = 2.92mm)
2. Compute of the remaining material depth on horizontal areas ≥ part offset + Min thickness on horizontal areas.

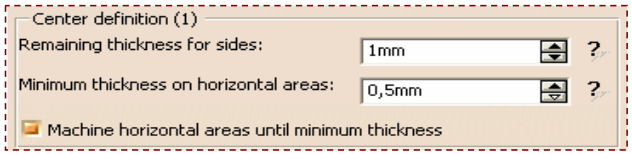
N : number of level
The zone 6 is not machined because there are upper top plane.

Student Notes:

Offset Management: Case 7- Mix Case

Parameters:

- Part offset =1mm (blue) → forbidden to go under this value
- Max depth of cut = 3mm
- Define Imposed plane with 0.5mm offset - - - - - - (Z=20.5)
- Define bottom plane with 1 mm offset - - - - - - - - (Z=11)



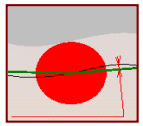
Condition to be respected:
Offset on each horizontal area = part offset + Min thickness on horizontal areas (1.5mm)

1. Recomputed depth to have regular depth of cut between imposed planes
until imposed plane: Recompute depth: H (top-imposed plane)/N closest than max depth of cut = 2.72 mm
after imposed plane: Recompute depth: H (imposed plane-bottom plane)/N closest than max depth of cut = 2.83 mm
2. Compute of the remaining material depth on horizontal areas = part offset + Min thickness on horizontal areas
The imposed plane path is done only in zones 1, 3 & 5. Bottom plane is done only in zone 5.
Machine horizontal area → 4 paths are done in different zones (1st: zone1, 2nd: zone2, 3rd: zone3, 4th: zone4, 5th: zone6)

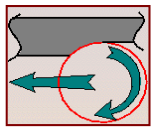
Student Notes:

Machining Tab (1/7)

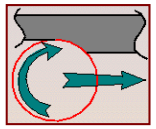
- Machining tolerance
  - Value of the maximum allowable distance between the theoretical tool path and the computed tool path.

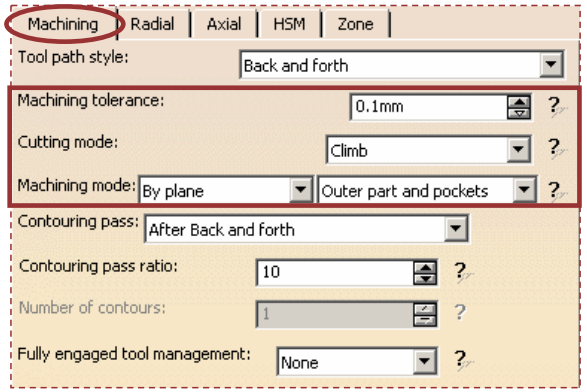


- Direction of cut definition:
  - Climb: The front of the advancing tool cuts into the material first



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





- Machining mode (refer to outer part and pocket area definition):
  This option allows to select geometry machining between-
  - Outer part and pocket,
  - Pockets only and
  - Outer part

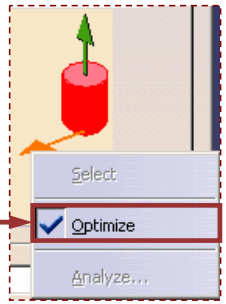
- Sequencing :
  - By plane or
  - By area

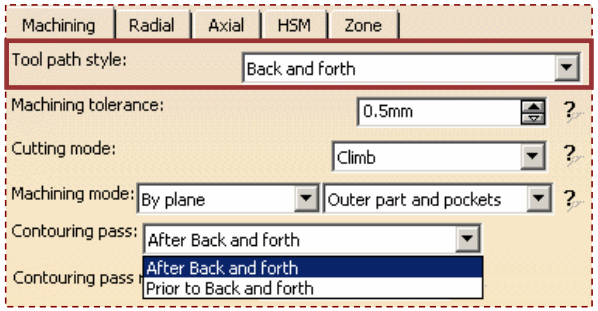
Student Notes:

Machining Tab (2/7)

Tool path style: Back and forth
Tool is moving following selected direction.
The machining direction is reversed from one path to the next.

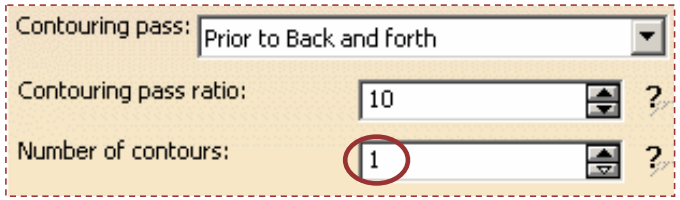
Optimize option let the algorithm choosing direction in order to minimize change of direction in tool path.

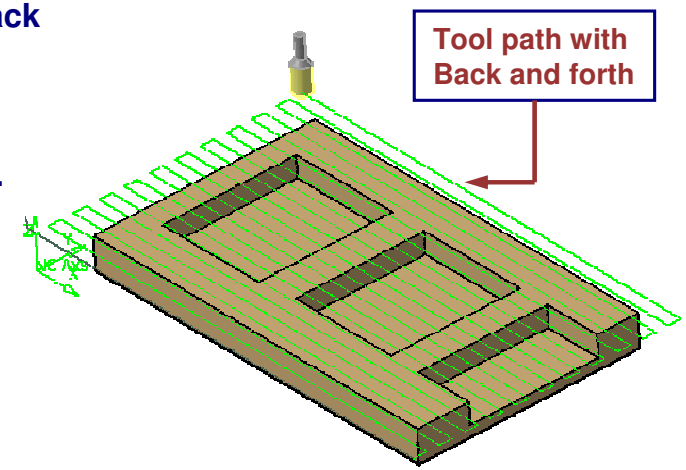




The contouring passes can be applied Prior or After the back and forth passes.

In 'Prior mode' it is possible to define a multi level contouring pass (in order to manage tool loading).



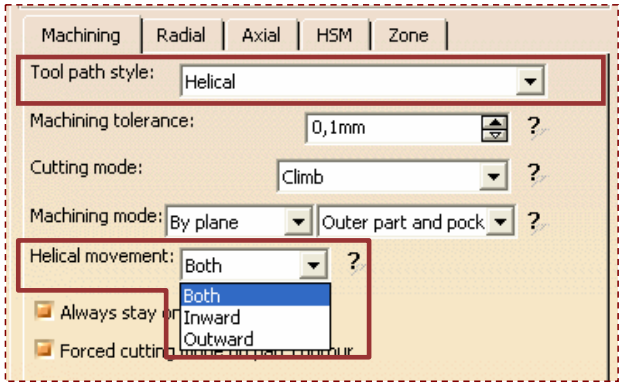


Student Notes:

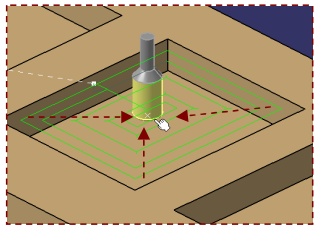
Machining Tab (3/7)

Tool path style: Helical

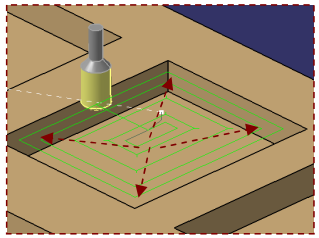
Tool moves in successive concentric passes from the boundary of the area to machine towards the interior or from the interior to the boundary.

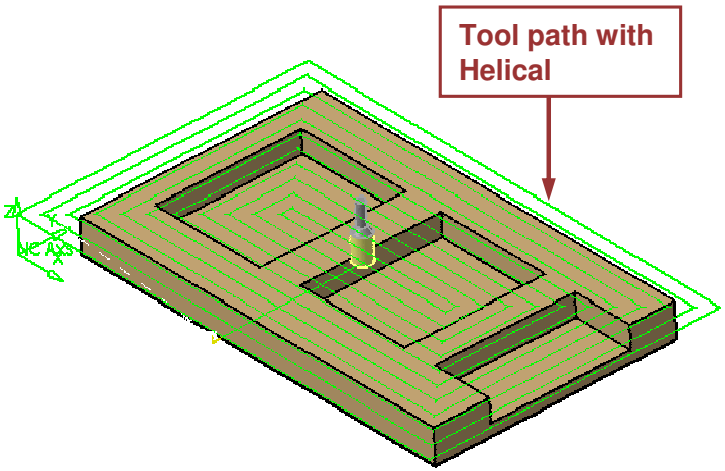


- **Helical Movement**:
  - **Inward:**
    Tools start from a point on zone boundary and follow concentric passes parallel to boundaries towards interior.



  - **Outward:**
    Tool starts from a point inside the zone and follow concentric passes parallel to boundaries.

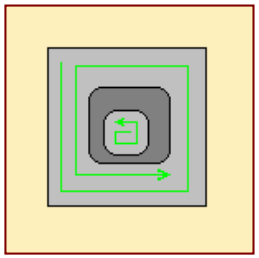


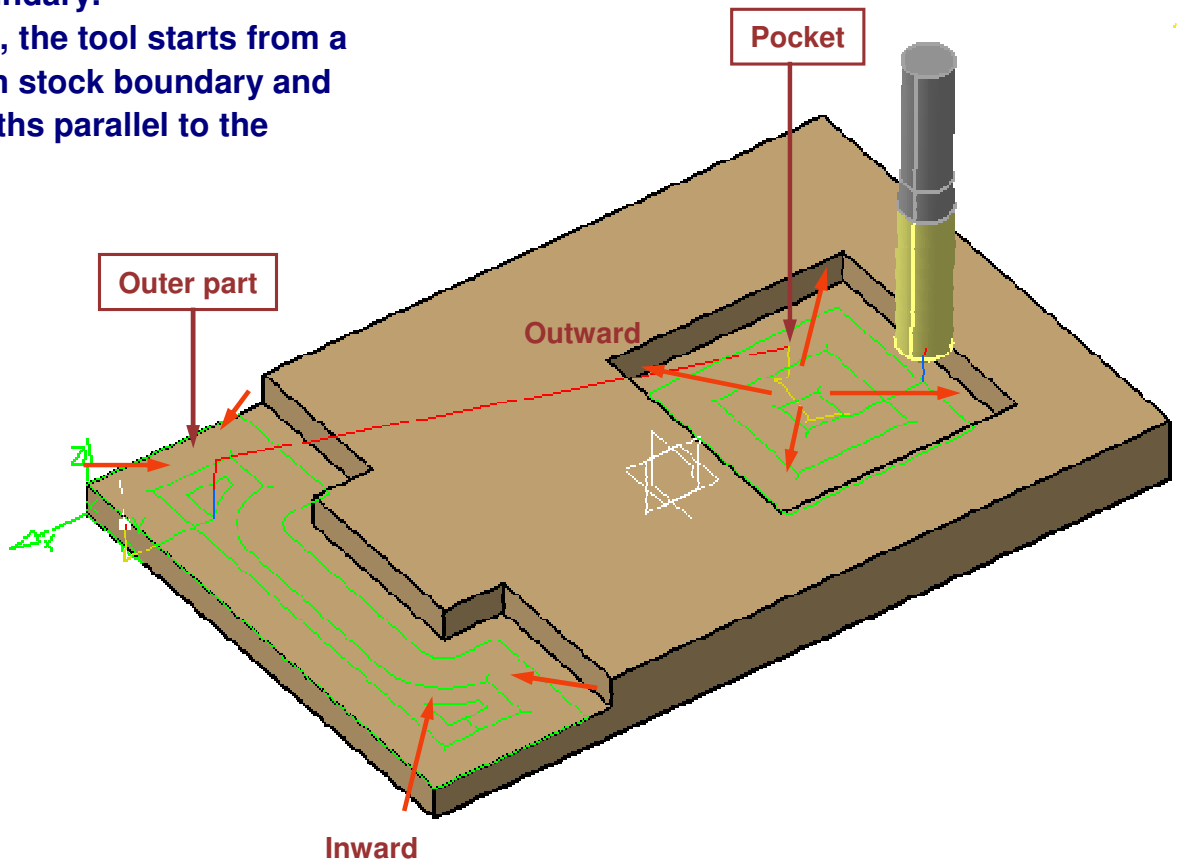


Student Notes:

Machining Tab (4/7)

- Both:
  - for pockets, the tool starts from a point inside the pocket and follows outward paths parallel to the boundary.
  - for external zones, the tool starts from a point on the rough stock boundary and follows inward paths parallel to the boundary.

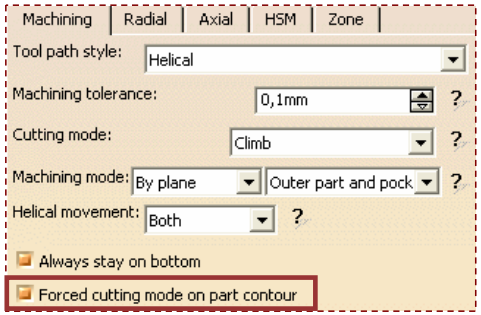




Student Notes:

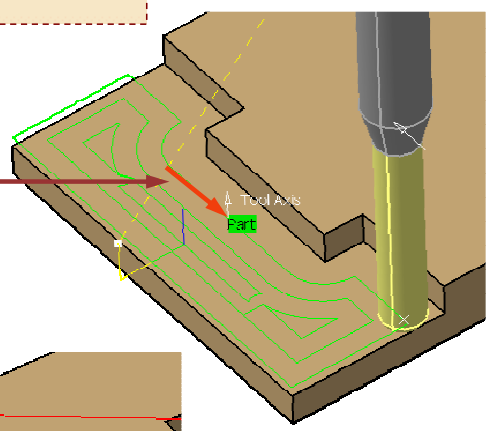
Machining Tab (5/7)

Forced cutting mode on part contour:



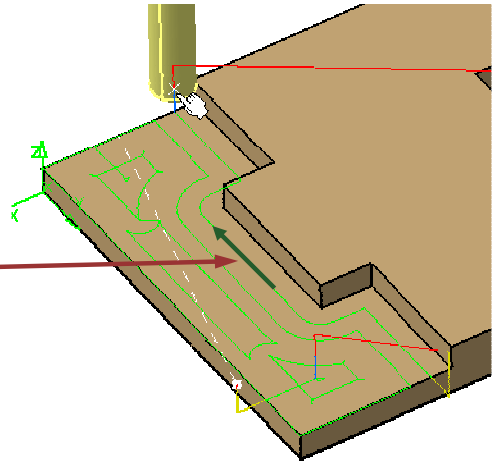
With 'Forced cutting mode on part contour' is deactivated, outer part in helical inward style, contouring pass is in Conventional cutting condition even if Climb cutting mode is selected.

Option OFF:
- **Inward**
- **contouring pass is not respecting climb**



With 'Forced cutting mode on contour' is activated, contouring pass is now in climb cutting condition. The tool goes round the outside contour of the part before continuing.

Option ON:
- **Inward**
- **contouring pass now in climb**

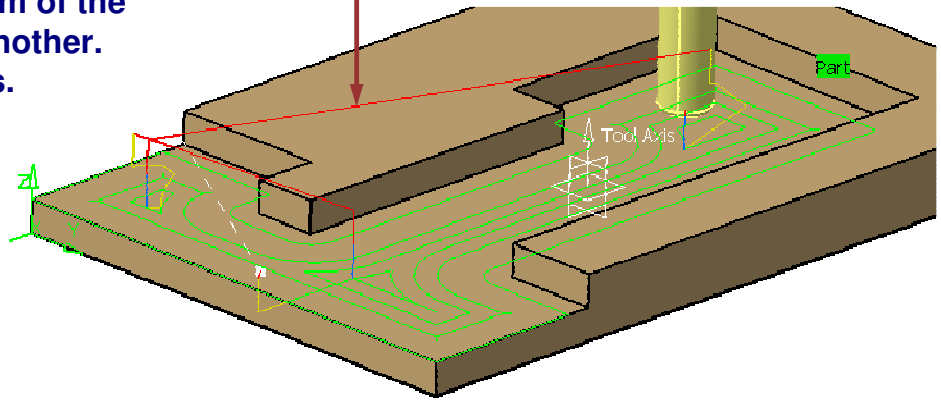


Student Notes:

Machining Tab (6/7)

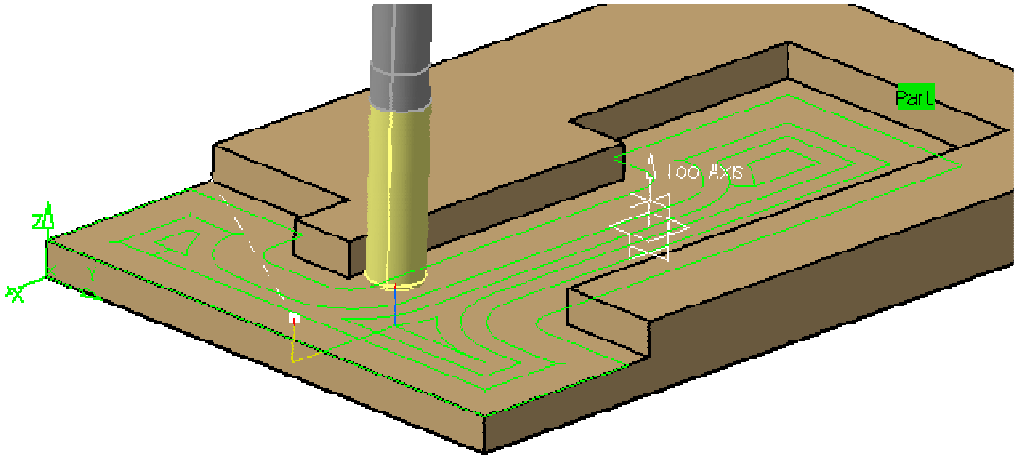
Always Stay on bottom:
It is possible when there is no collision and with tool staying in the machining plane.
The tool to remain in contact with the bottom of the pocket when moving from one domain to another.
This avoids unnecessary linking transitions.

Linking transitions



Option OFF:

- Approach macro
- Retract macro
- 2 linking movements



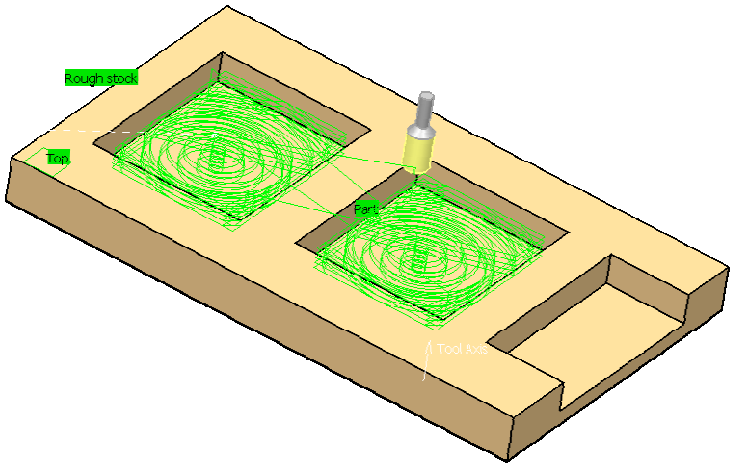
Option ON:

- Approach macro
- Retract macro
- No linking movements

Machining Tab (7/7)

Tool path style: Concentric

- Tool is moving following concentric passes.
- Tool removes the most constant amount of material possible at each concentric pass.
- Tool is never fully engaged in material.
- Tool path is always respecting given cutting mode.
- Approach macro is only helix one.



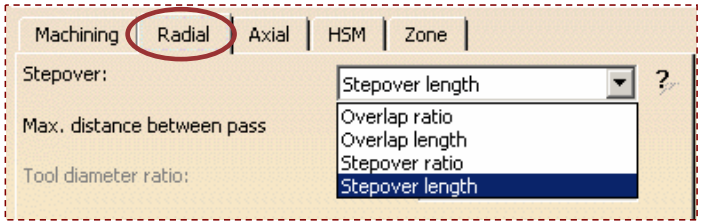
Student Notes:

Student Notes:

Radial tab

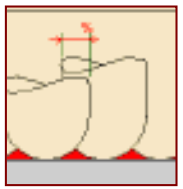
There are four different ways to define distance between passes:

- Overlap ratio
- Overlap length
- Step over ratio
- Step over length

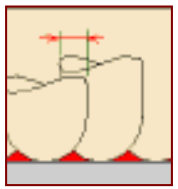


Overlapping:

Overlap ratio:
It is the overlap between two passes, given as a percentage of the tool diameter.

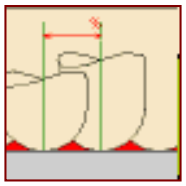


Overlap length:
It is the distance between two passes with respect to a tool diameter ratio recovery.

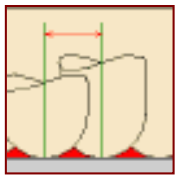


Stepover

Stepover ratio:
It is the stepover between two passes, given as a percentage of the tool diameter.



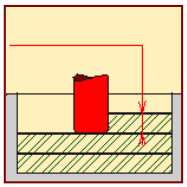
Stepover length:
It is the maximum distance between two passes.

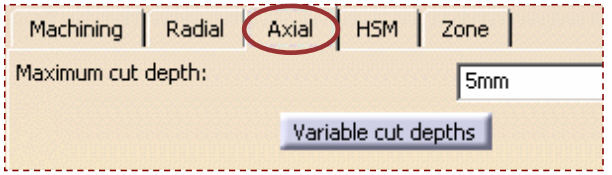


Student Notes:

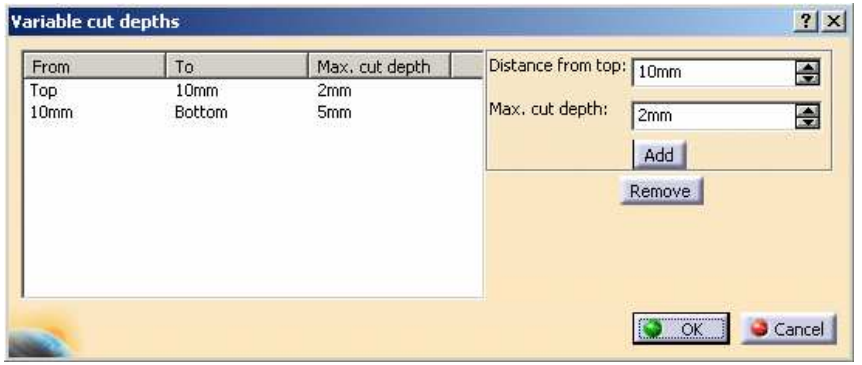
Axial tab

Maximum cut depth:
It defines the maximum depth of cut per axial level.
This value will be respected for each axial level from top to bottom plane.





Variable cut depths:
It allows to define different values of maximum depth of cut depending on axial levels.



Student Notes:

HSM tab

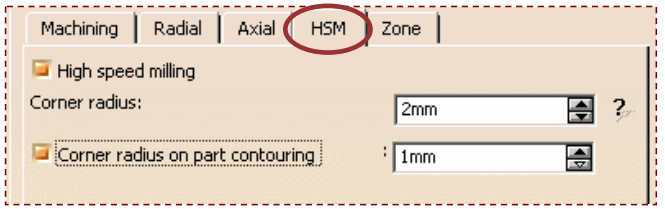
High Speed Milling technological parameter:

In order to be compliant with machine technology, this parameter allows to avoid corners in toolpath, by defining the minimum radius of tool path.

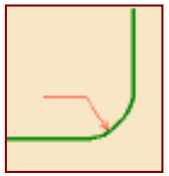
It is possible to have a different cornerization on part contouring (most of the time a smaller one to reduce rework).

Center cornerization is linked with ‘step over distance’.

A warning message as shown during Tool path computation is raised in case of incompatibility and if the value is set at maximum.



Corner radius:
It defines the radius of the rounded ends of passes. The ends are rounded to give a smoother path that is machined much faster. The corner radius is not applied to the finish path.



Corner radius on part contouring:
It specifies the radius used for rounding the corners along the Part contouring pass of a HSM operation. This radius must be smaller than Corner radius value.

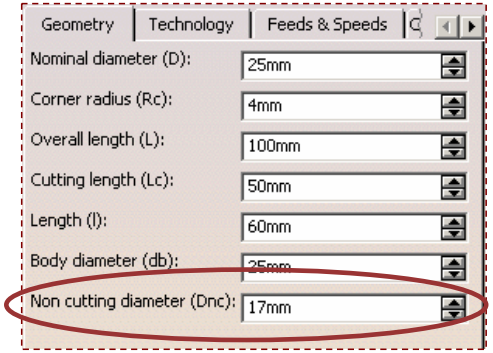
Student Notes:

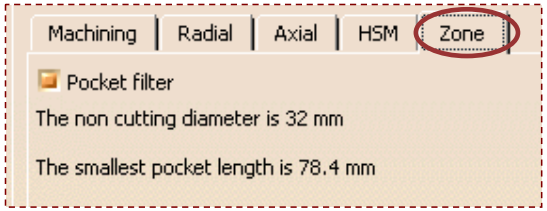
Zone tab

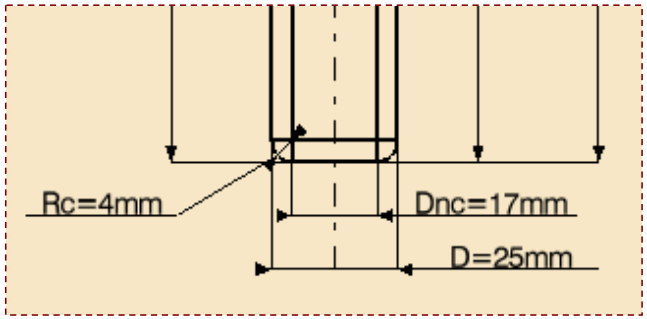
Zone definition:

This parameter is acting like a 'pocket filter', which means small pockets will be removed.

To be activated one must define a "non- cutting diameter (Dnc)" parameter in tool description.







Based on this value the following formula is applied to define the smallest machinable pocket length:

XX(mm) = Dnc+D+2 x (machining tolerance)

There will not be machining path in pockets where tool can't plunge without respecting maximum plunge angle.

Student Notes:

To Sum Up

In this section you have seen:

- **Necessary geometrical elements to define a Cavities Roughing operation**
  - **Parts (can be composed of different elements)**
  - **Stock**
  - **Planes (top, bottom, imposed)**
- **Machining strategies of Cavities Roughing**
  - **Helical, Back and Forth, both with HSM option**
- **Radial and Axial strategies**