Student Notes:

CATIA V5 Training
Foils

Multi-Pockets Machining

Version 5 Release 19
January 2009

EDU_CAT_EN_MPG_FF_V5R19

Student Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to define Multi-Pockets Operations:

- **Power Machining**
- **Multi-Pockets Flank Contouring**

Targeted audience

NC programmer knowing how to work with CATIA V5 Parts and already skilled in Machining product

Prerequisites

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

4 hours

Student Notes:

Table of Contents (1/2)

Student Notes:

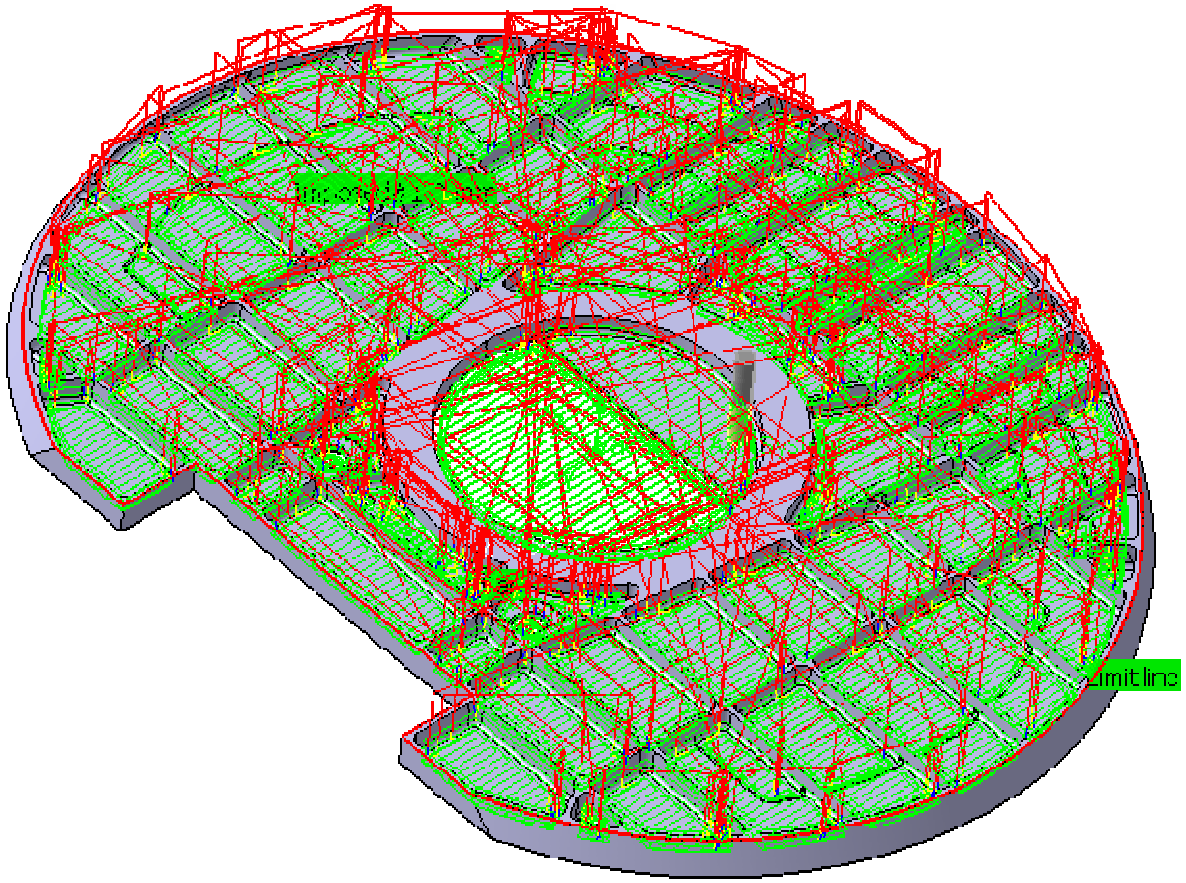
Table of Contents (2/2)

STUDENT GUIDE

Student Notes:

Introduction to Multi-Pockets Machining

You will become familiar with the Power Machining and Multi-Pockets Flank Contouring principles.



Student Notes:

About Multi-Pockets Operations

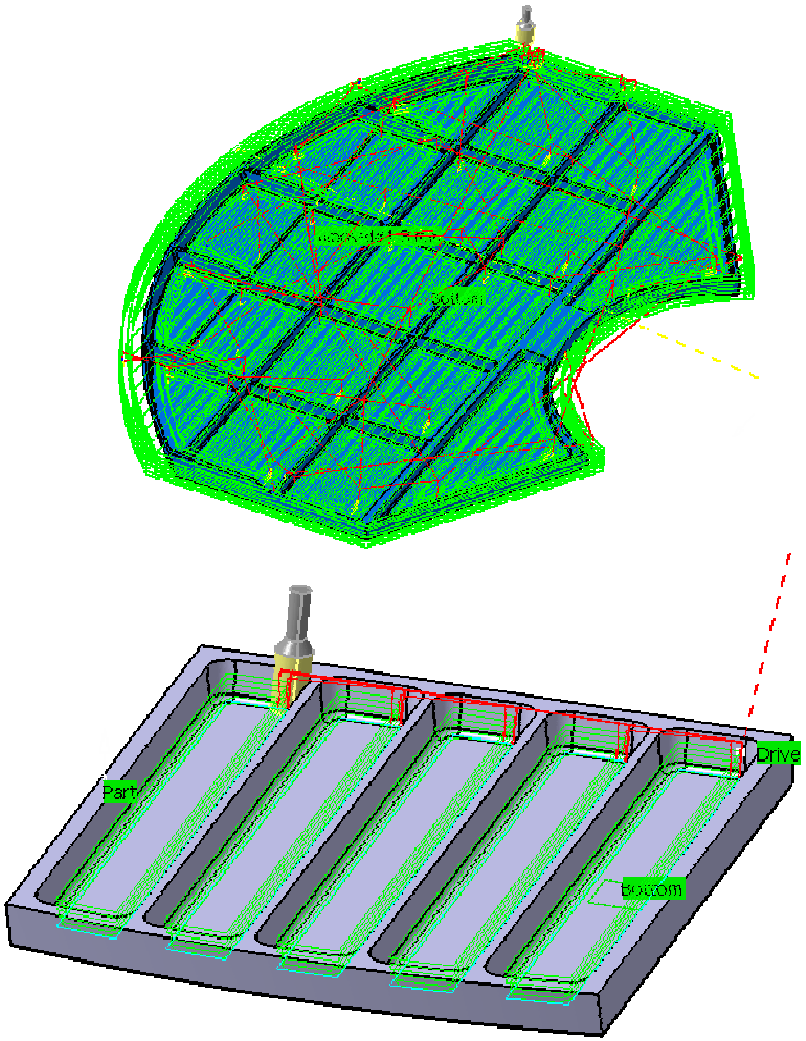
The Multi-Pockets operations provide you highly productive global rough to finish machining strategies.

MPG delivers a process focused solution to machine multi-cavity parts such as structural prismatic parts or aerospace structural parts.

This breakthrough solution delivers to NC programmers the possibility to machine multi-cavity parts with a mix of roughing and finishing tool paths.

The Multi-Pockets operations enable to machine the part in a global and automatic way that drastically reduces the programming time.

Multi-Pockets Flank contouring operation is useful for the machining of part having multiple pockets with an obvious view direction.



Student Notes:

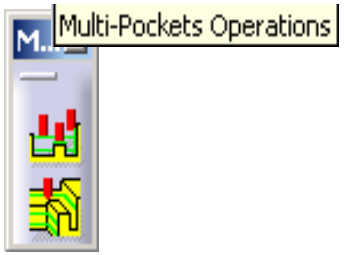
How to Access Multi-Pockets Operations

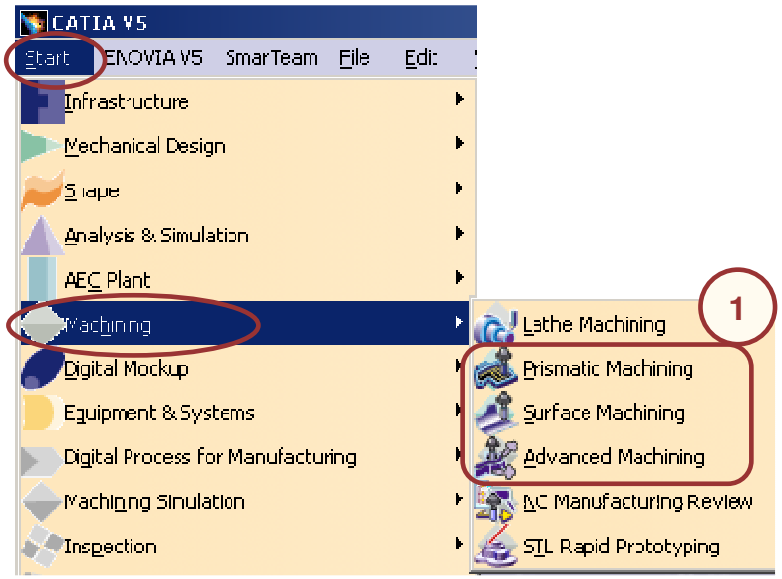
1. **Menu > Start > Machining >PMG/ SMG/ AMG**

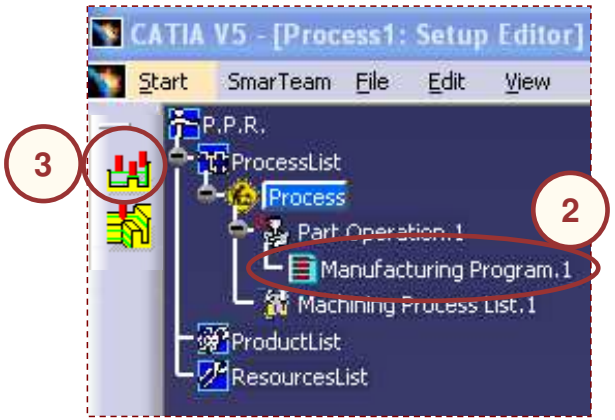
   MPG is an add-on product to all milling products (PMG, SMG, AMG)

2. **Select Manufacturing Program.1**

3. **Select Power Machining Icon in Multi-Pocket Toolbar**







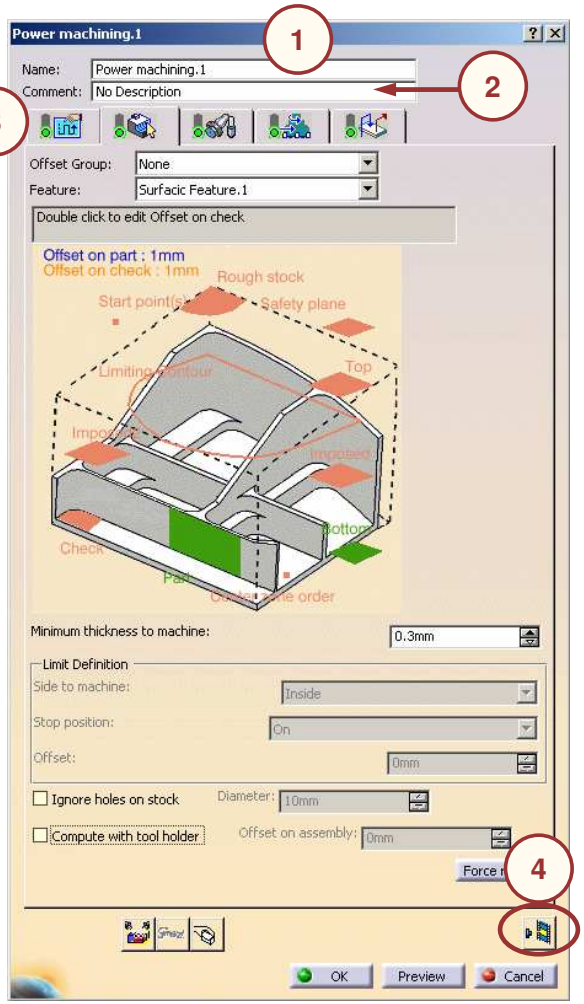
Student Notes:

Creating a Multi-Pockets Operation: General Process

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

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

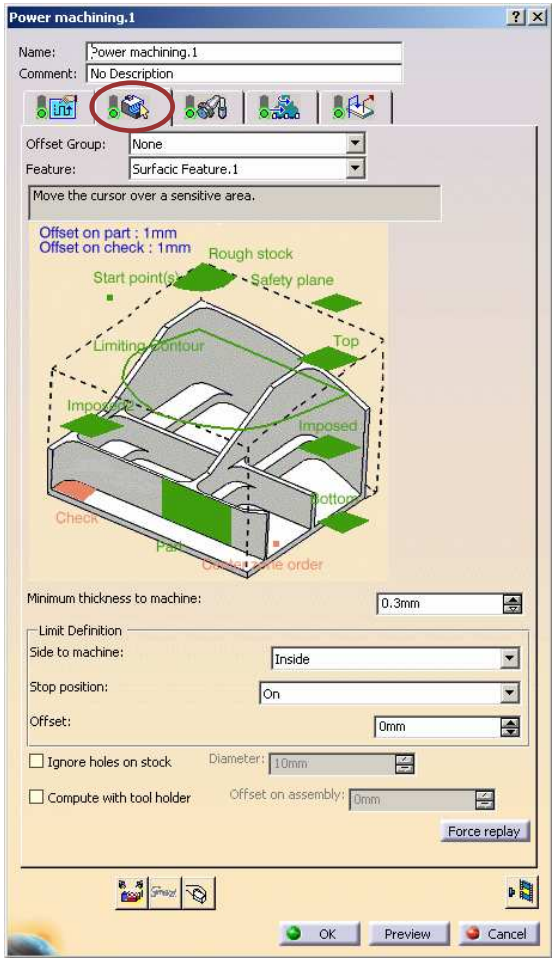
4. Replay and/or Simulate the operation tool path



Student Notes:

Power Machining: Geometry

You will see the options in the Geometry tab of Power Machining.



Presentation (1/2)

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

- **A and B : Rough stock and Part**

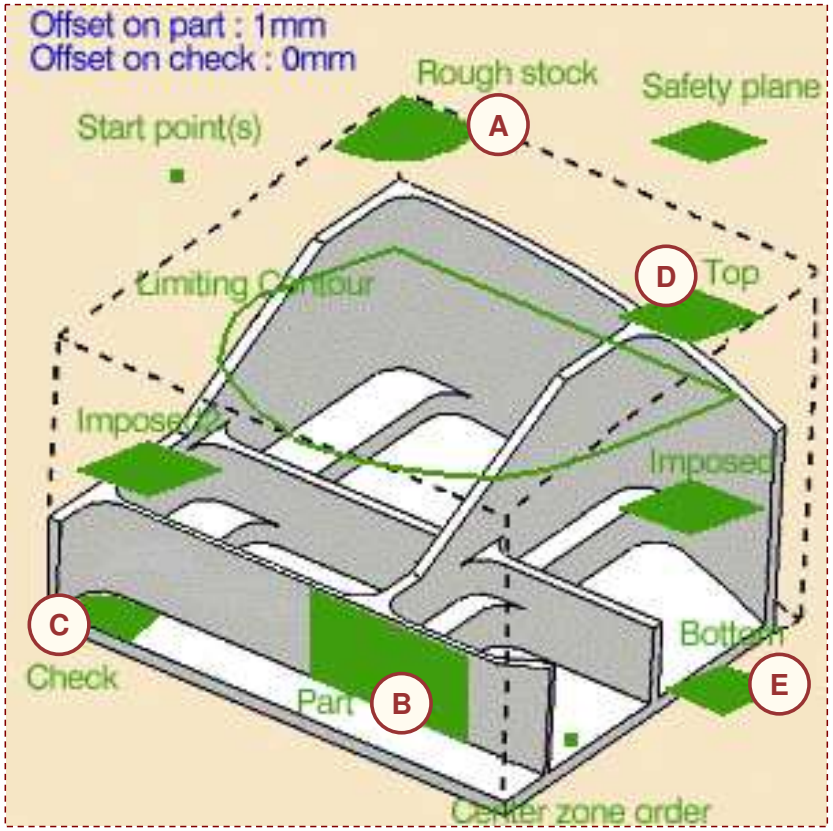
  Multi-pocket operation will remove all stock material in order to obtain final part. Offset can be applied on part.

- **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:

Student Notes:

Presentation (2/2)

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

- **F : Imposed planes (two groups)**

  Force cutter to machine in this plane (global offset can be applied on each group)

- **G : Limiting contour**

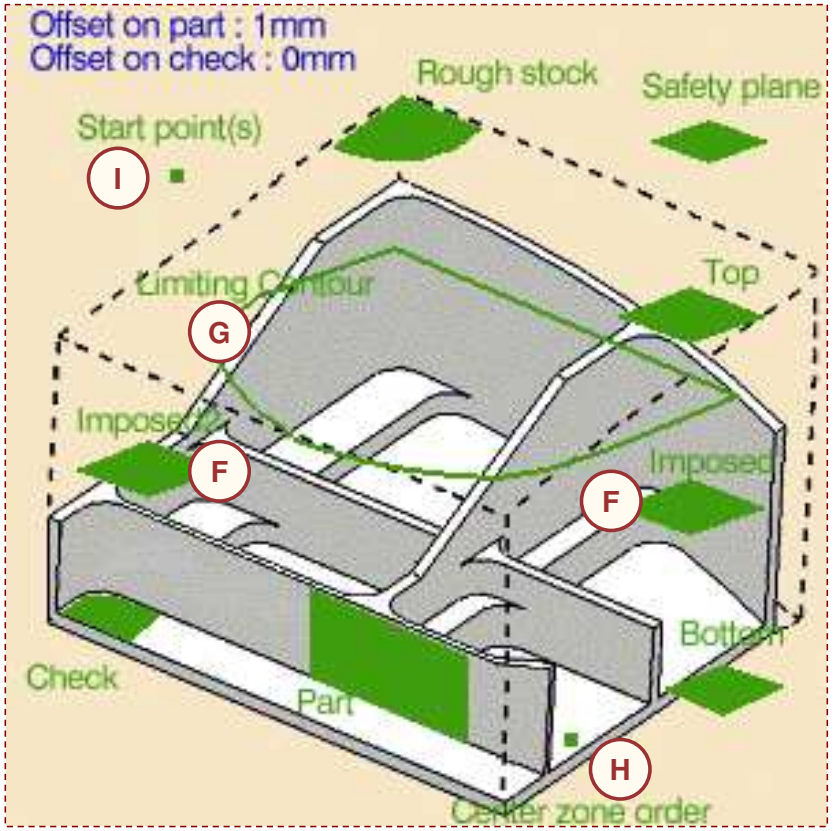
  Re-limit machining area after stock and part definition

- **H : Center zone order**

  Define pocket machining order

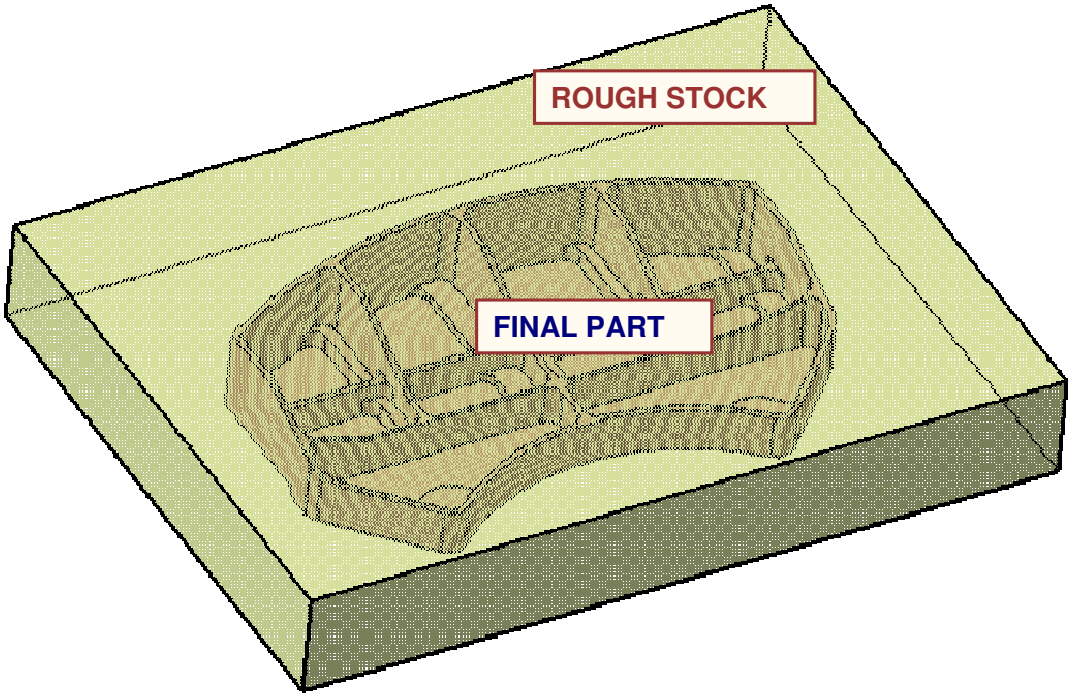
- **I : Start Point (optional)**

  Impose start point in open area (not in pocket)



Geometry Parameters (1/9): Rough Stock and Part

Rough Stock and Part definition example:



Student Notes:

Student Notes:

Geometry Parameters (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.

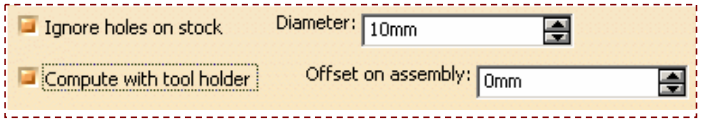
It is recommended to use helical strategy for rework computation in order to have an optimized toolpath.

Minimum thickness to machine parameter:

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

Ignore holes on stock:

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



Compute with tool holder:

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

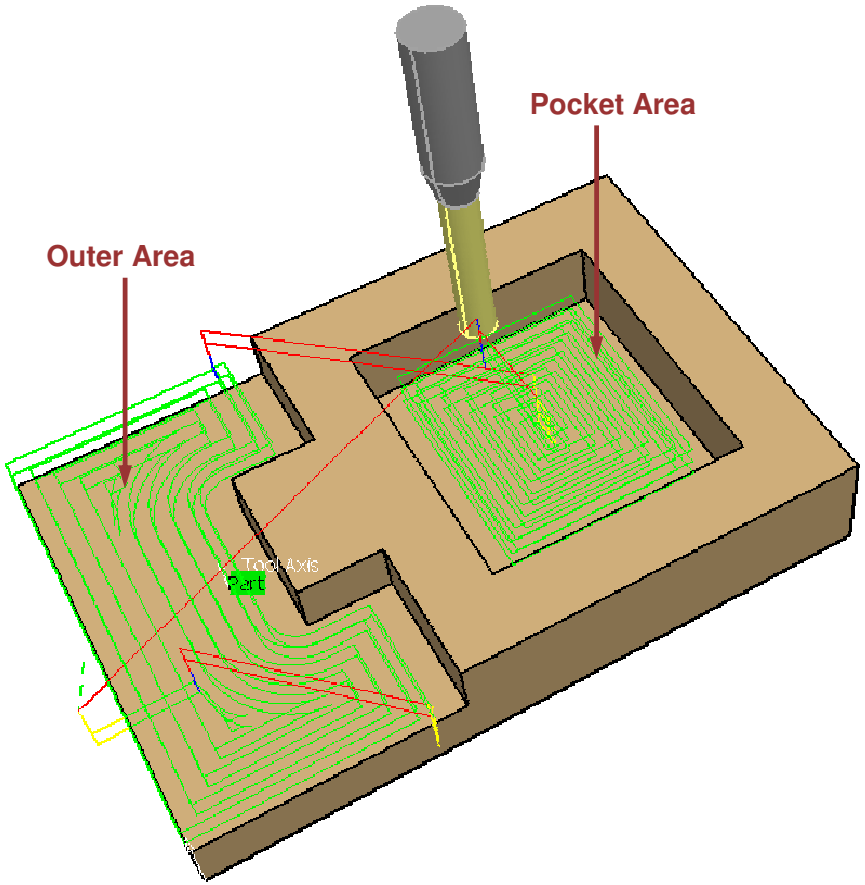
Student Notes:

Geometry Parameters (3/9): Outer Area 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.**

  Depending on which z level plane the area is analyzed

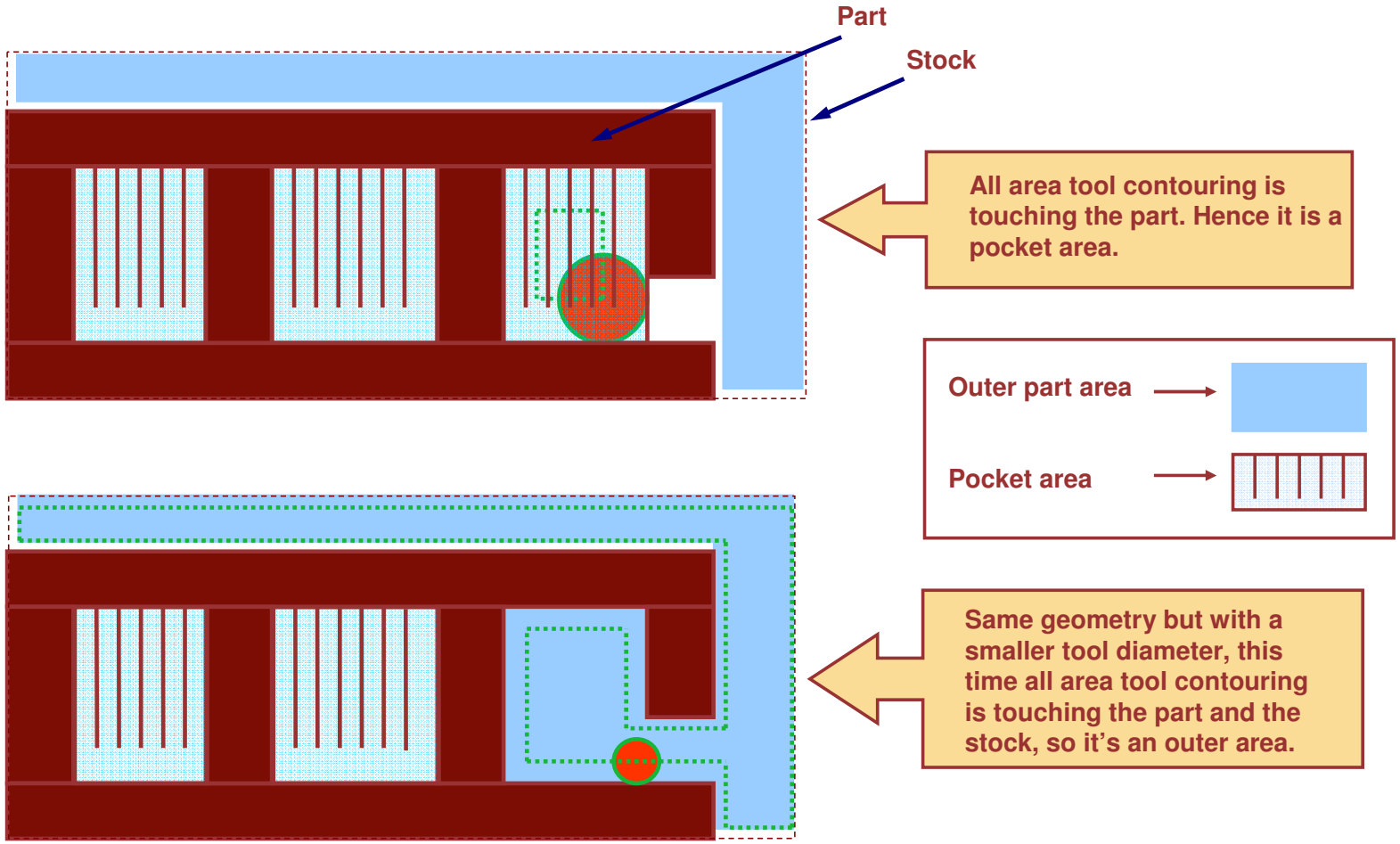
- **Outer part and pocket notes:**
  - **It is not only a geometrical concept. It is a function of: Part, Tool diameter and Stock.**
  - **Part can be composed of different elements depending of tool diameter. A pocket can become an outer part.**



Student Notes:

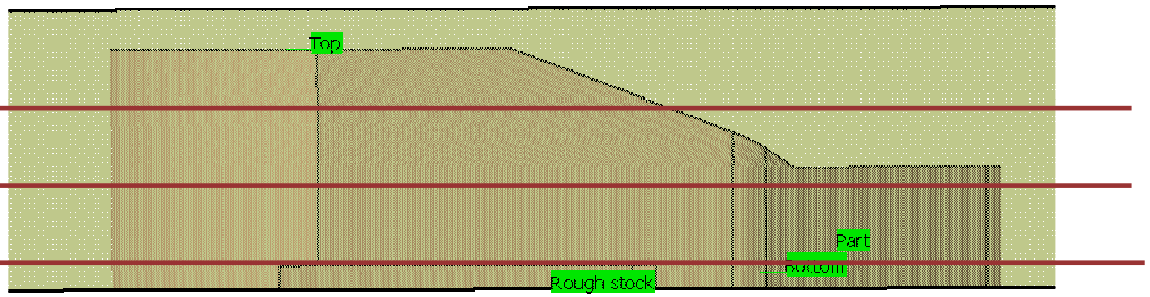
Geometry Parameters (4/9): Important Note

Tool diameter impact on outer and pocket area:



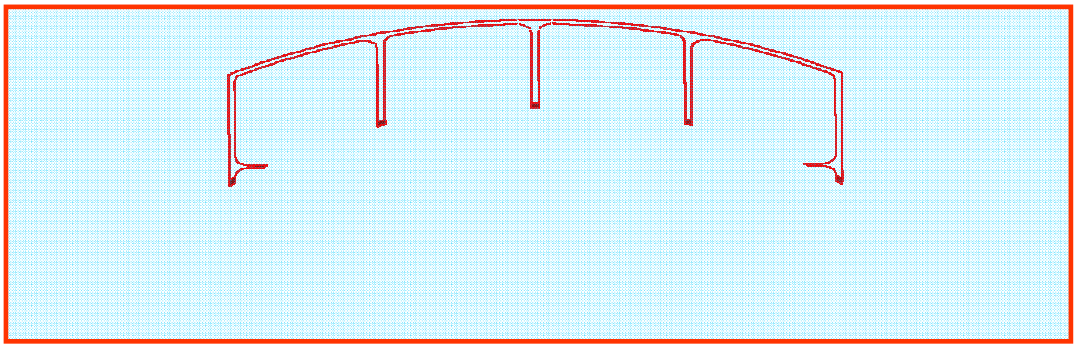
Geometry Parameters (5/9): Z Level Plane Impact on Area

Example:



Cut plane 1

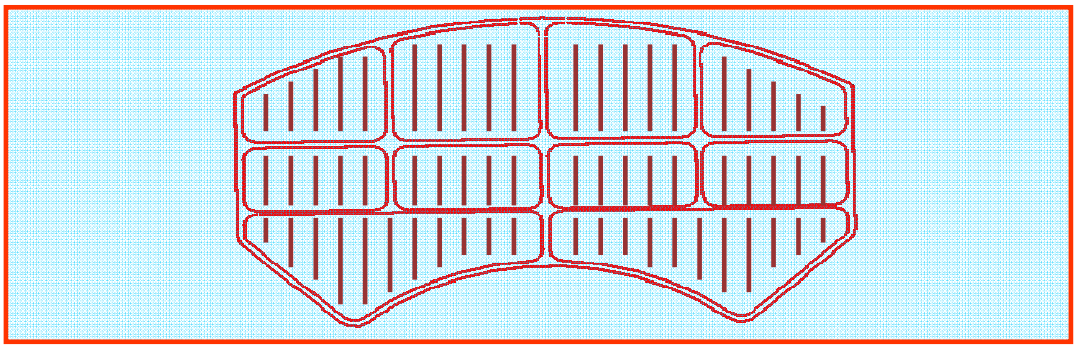
Outer part area



Cut plane 2

Outer part area

Pocket area

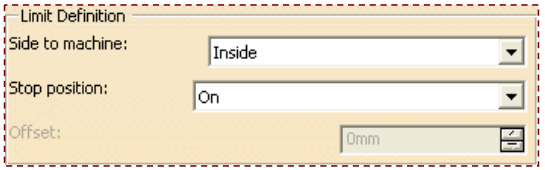


Student Notes:

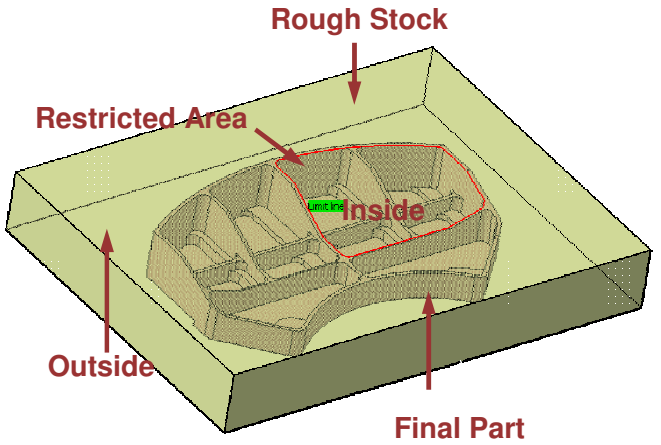
Student Notes:

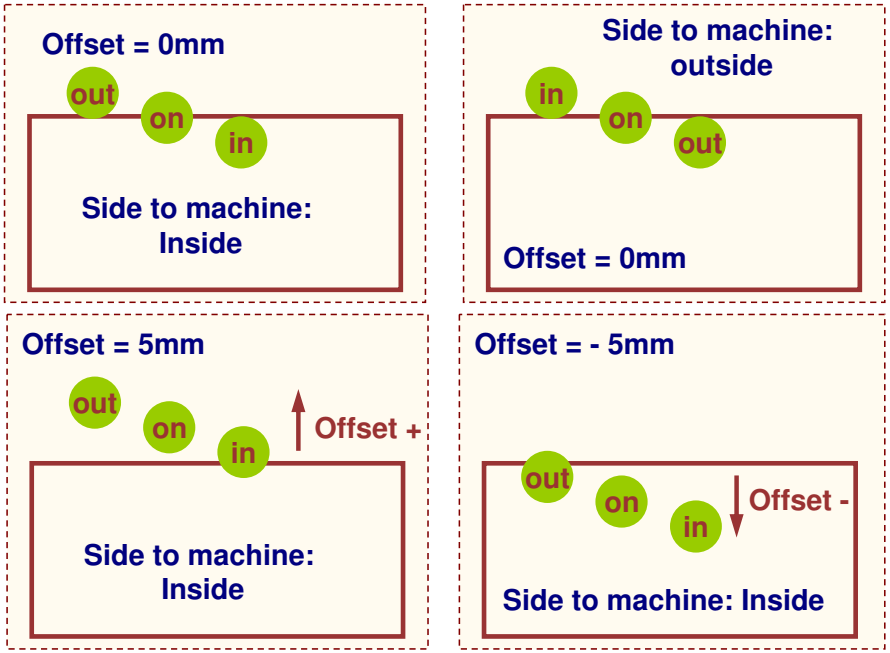
Geometry Parameters (6/9): Limiting Contour

Limiting contour is used to restrict machining area to dedicated pockets. You 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).





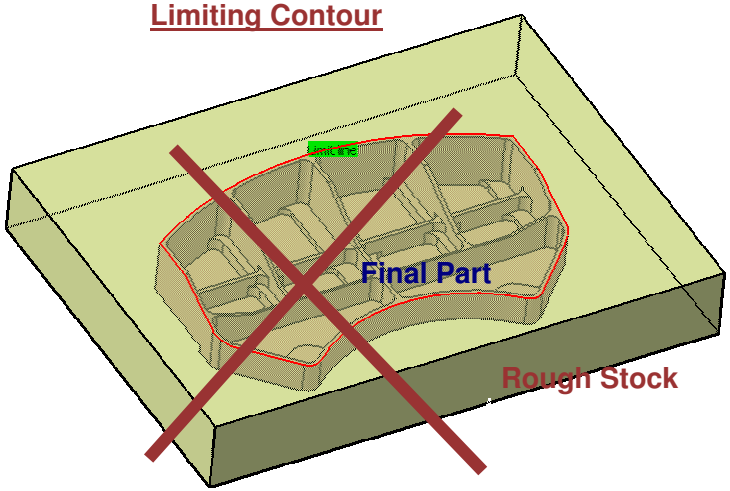
Positive Offset => offset to outside
Negative Offset => offset to inside

Student Notes:

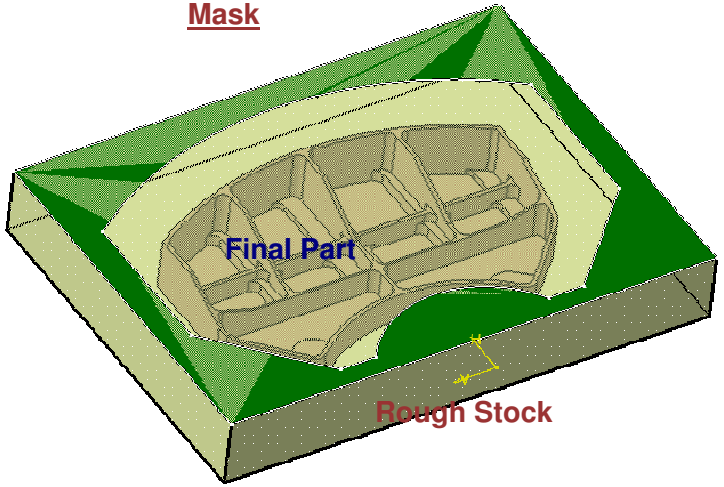
Geometry Parameters (7/9): Mask Methodology

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 a part in the user interface.

Limiting Contour



Mask



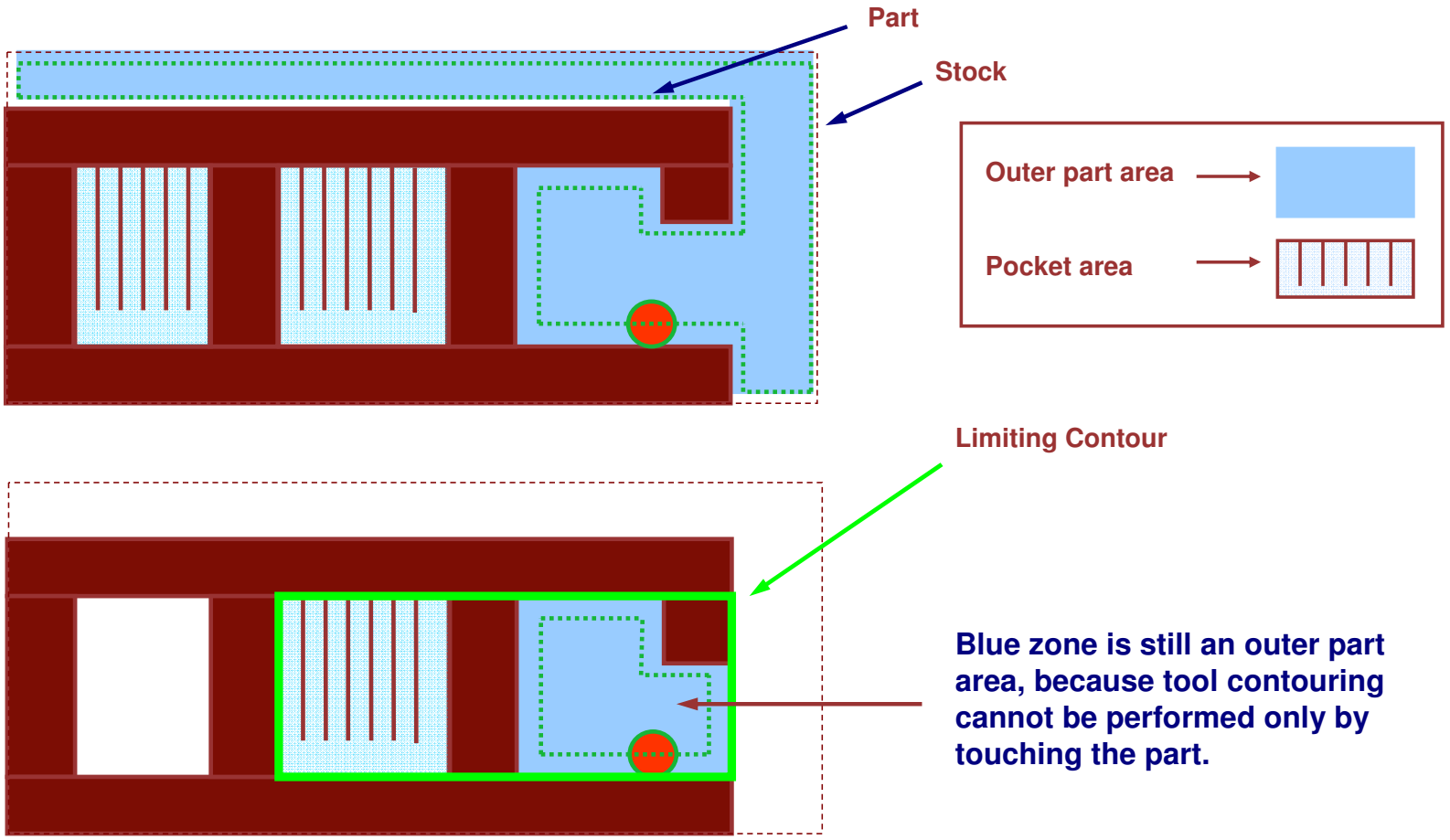
Negative points:
Limiting contour can be crossed by tool tip
Limiting contour impact on outer and part area
Need to manage offset
Offset is function of Tool diameter, thickness on part.

Positive points:
Part elements are not necessarily connected.

Student Notes:

Geometry Parameters (8/9): Important Note

Limiting Contour impact on outer and pocket area:

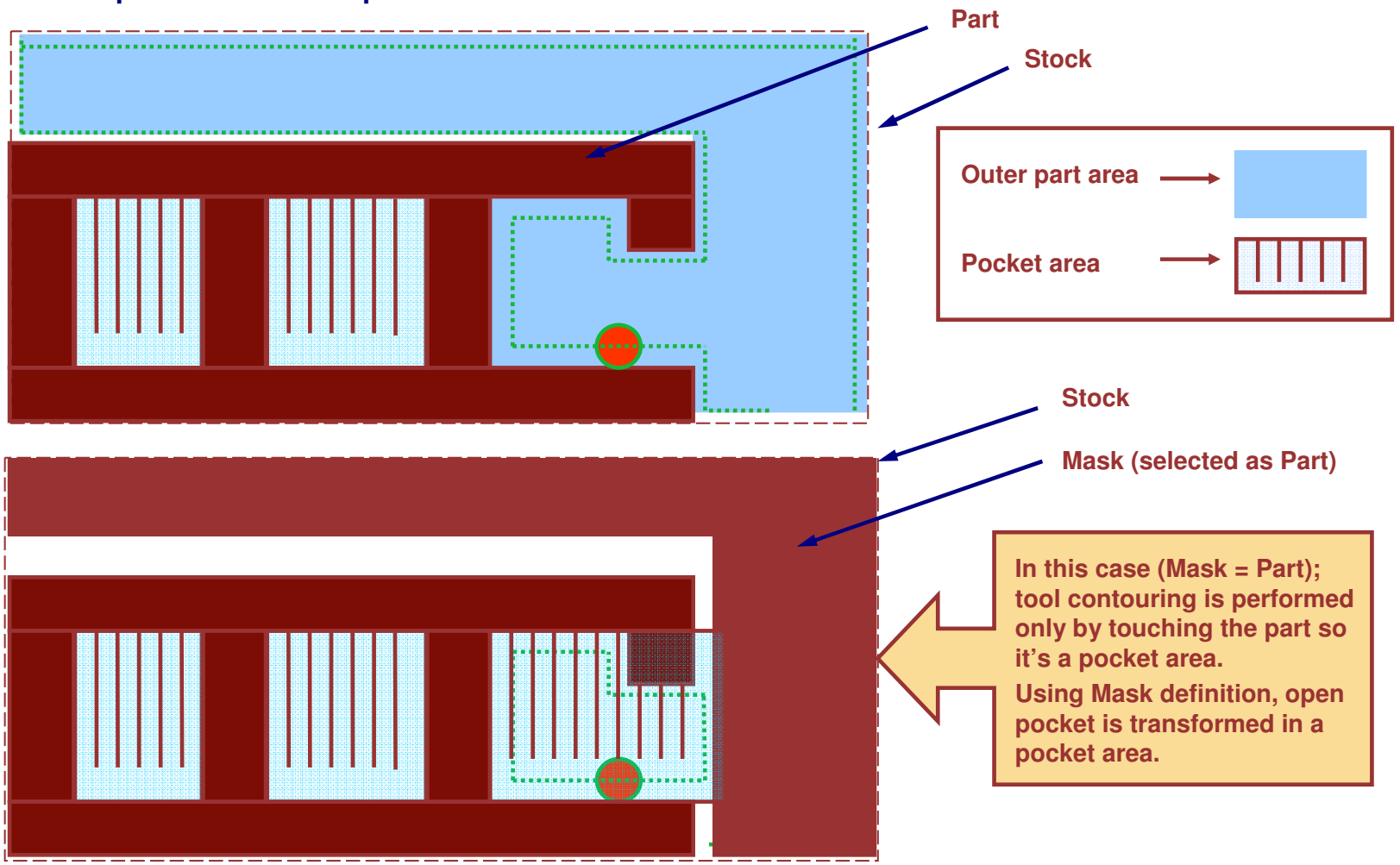


Blue zone is still an outer part area, because tool contouring cannot be performed only by touching the part.

Student Notes:

Geometry Parameters (9/9): Important Note

Mask impact on outer and pocket area:

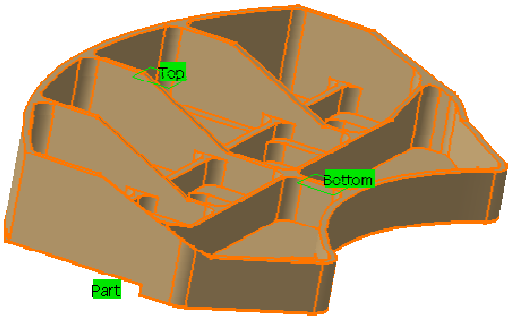


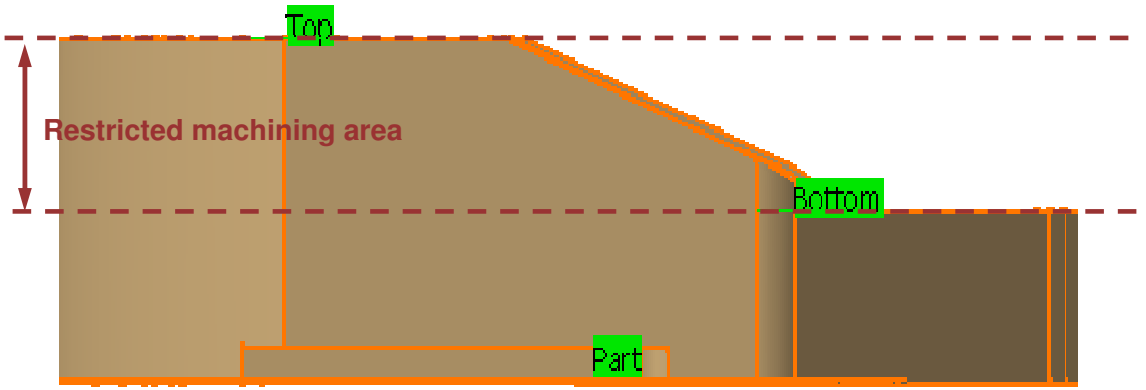
Student Notes:

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 section)



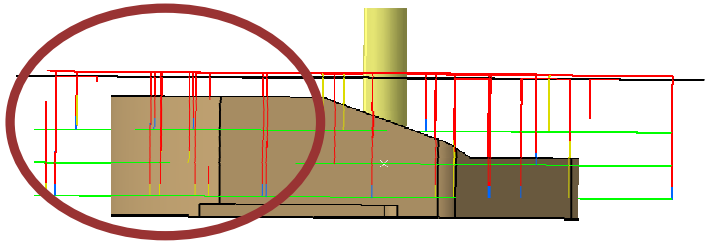


Student Notes:

Planes (2/4): Imposed Planes

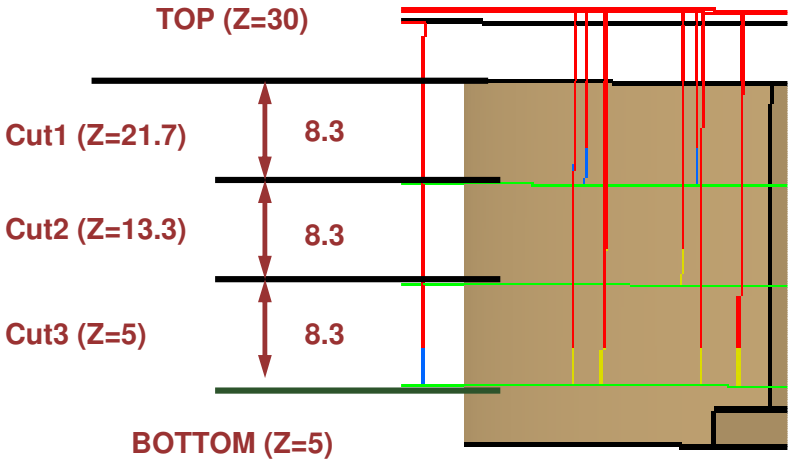
Top and bottom planes with maximum depth of cut allow to define cutting planes.

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

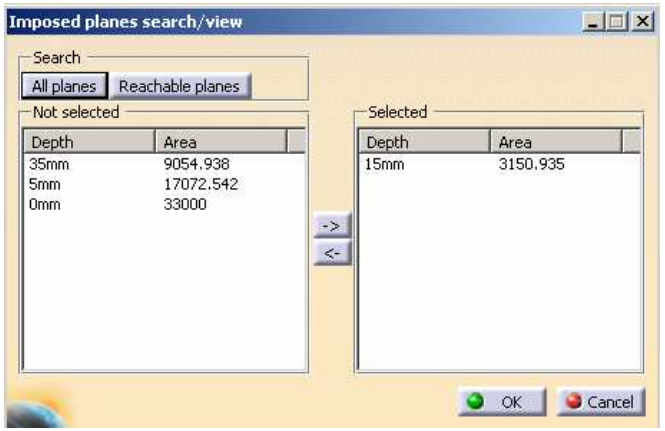


Student Notes:

Planes (3/4): Imposed Planes

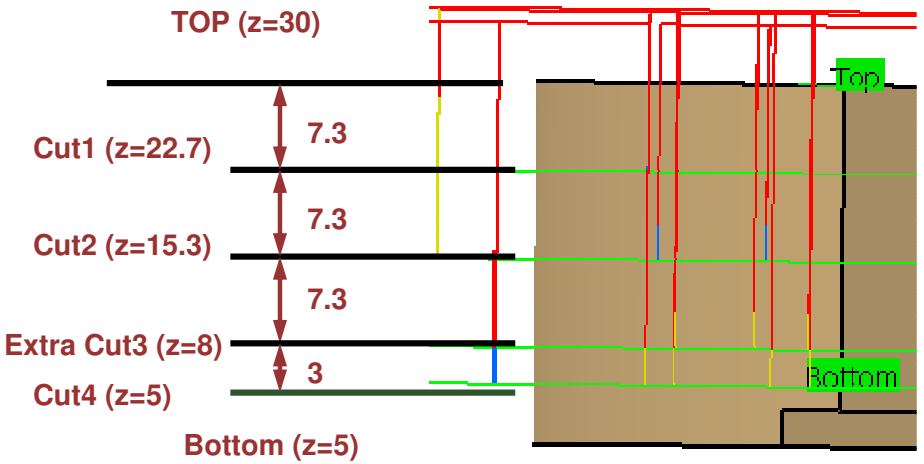
Adding imposed plane with search plane capability:

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.

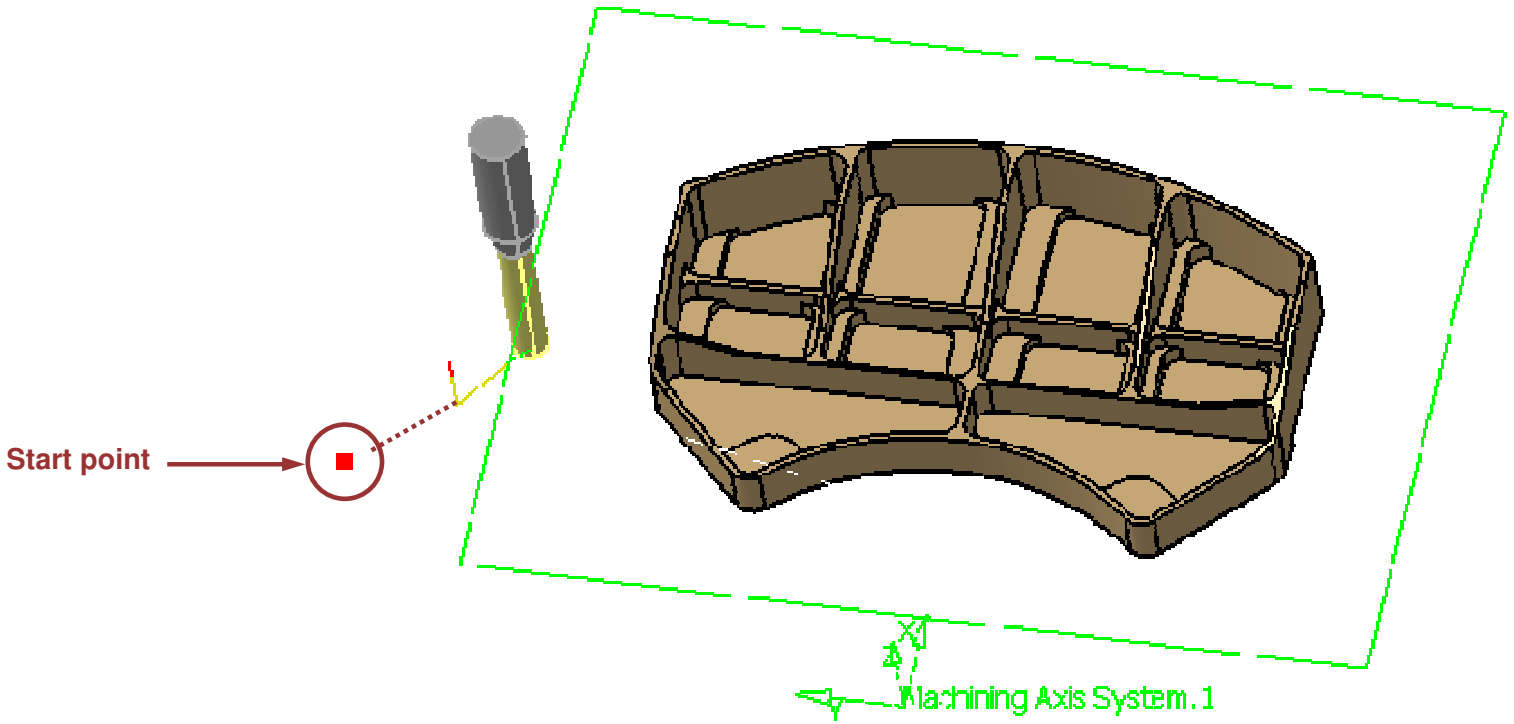
Adding imposed plane manually:
Any plane can be selected (physical part plane, plane created in WFS workbench etc)

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 (1/2)

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



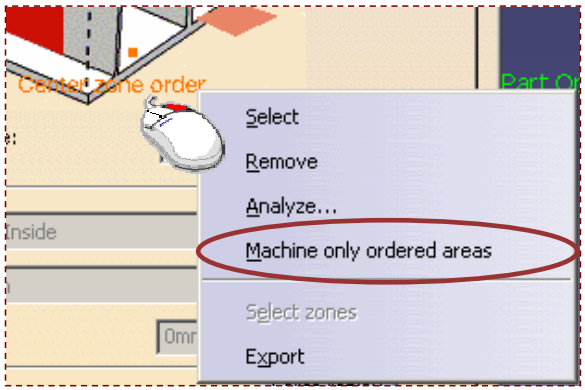
Student Notes:

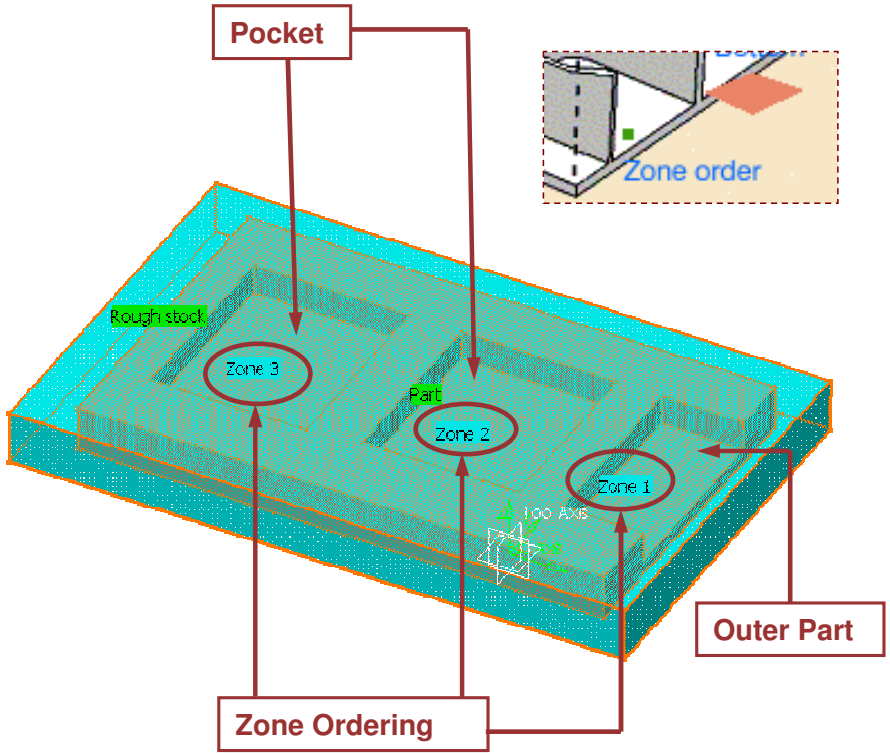
Start Point and Zone Order (2/2)

- **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.
(Right-click on zone order)

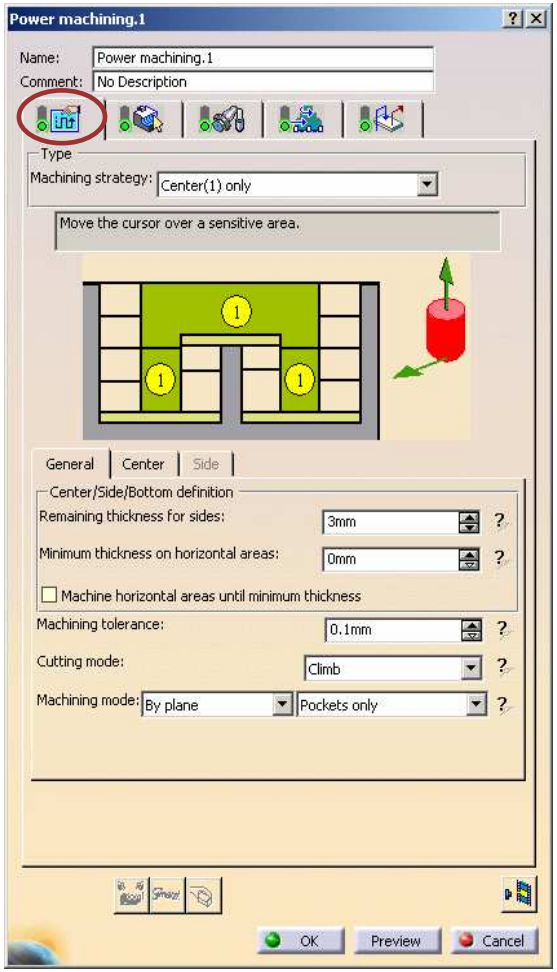




Student Notes:

Power Machining: Strategy

You will learn the options in the Strategy tab of Power Machining.



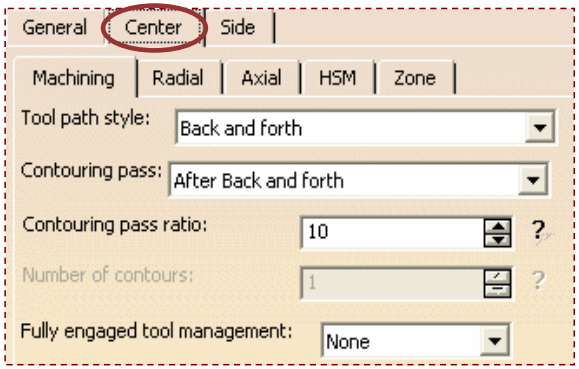
Student Notes:

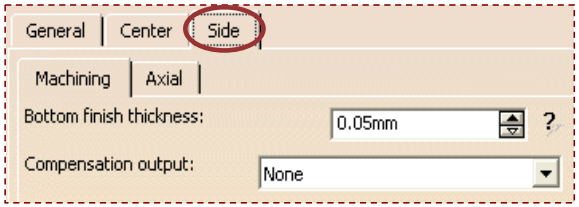
Presentation

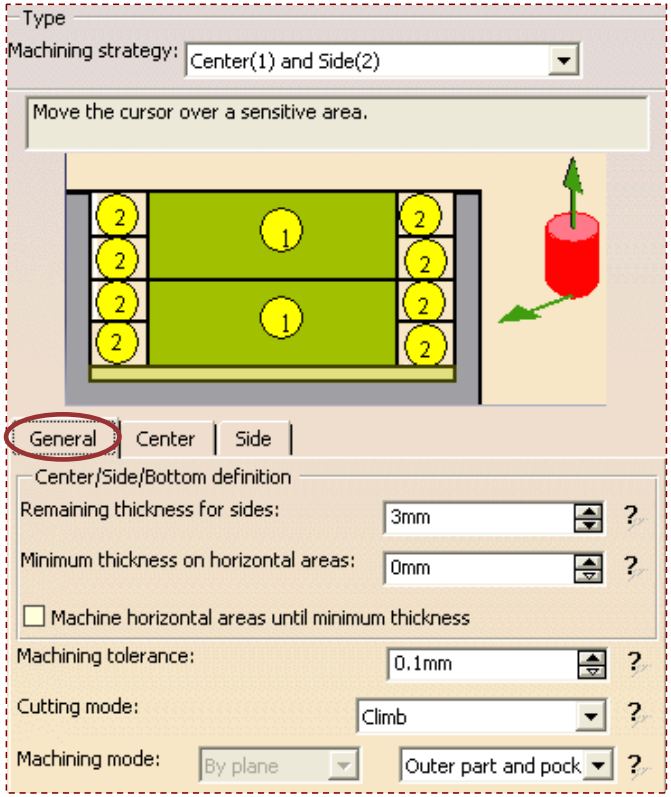
This tab Page allows you to define

- General parameters (which are common to Center and Side)
- Thickness, Machining tolerance and Mode

The two separate tabs allow to define dedicated Center and Side parameters:



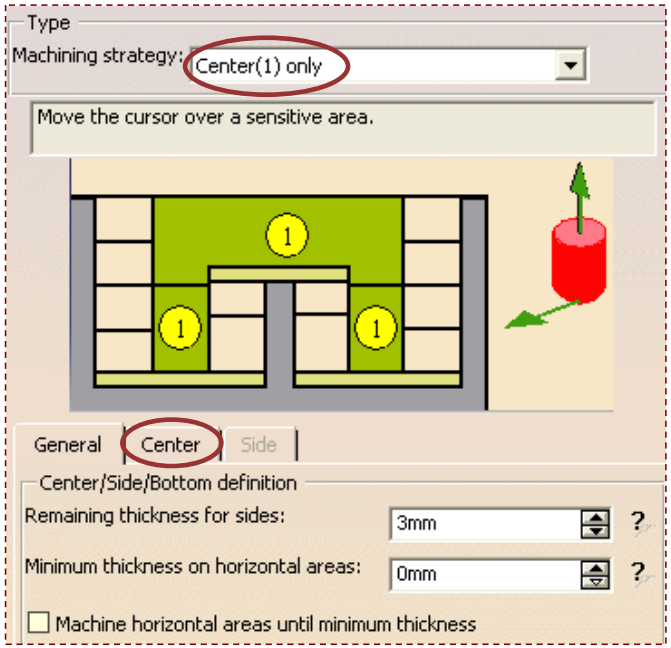




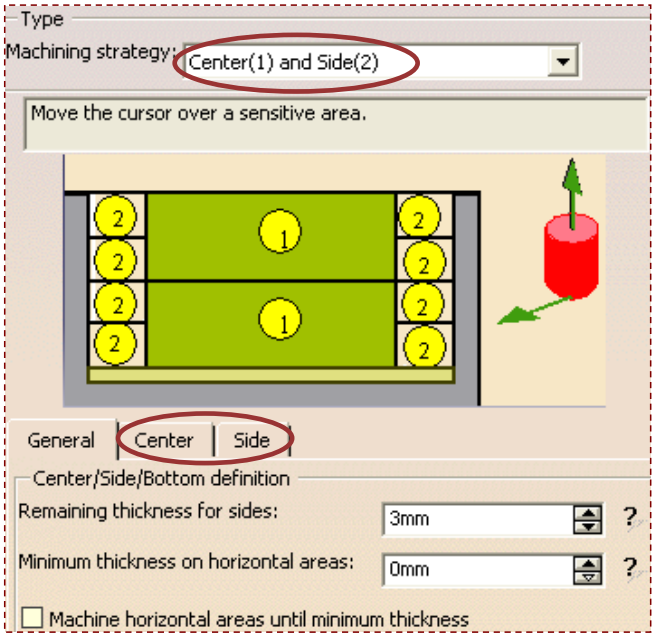
Student Notes:

Machining Strategy Type

Power machining operation allows you to define Center and Side tool path in a single operation. It is possible to define only Center parameters in both types.



Center only strategy selected. You need to define only Center parameters.



Center and Side strategy selected. You need to define both Center and Side parameters.

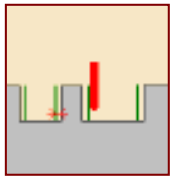
Student Notes:

General Parameters (1/5)

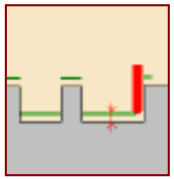
Power Machining is rough machining of the part by keeping thicknesses on sides and horizontal areas.

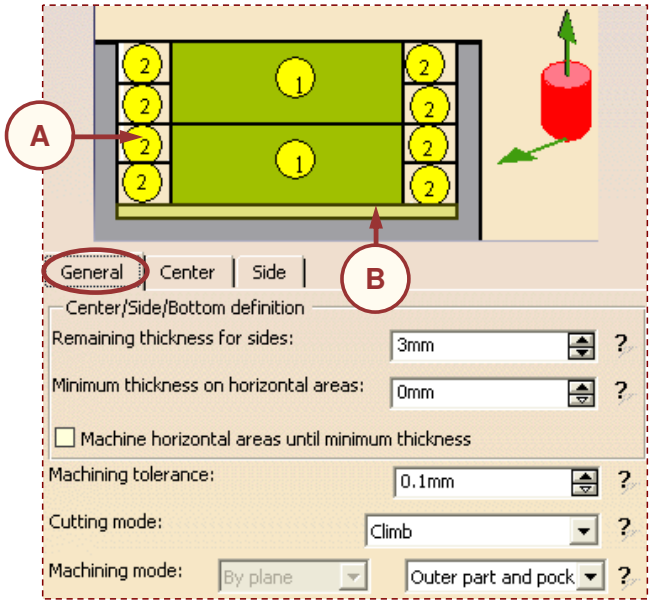
You can define:

A. Remaining thickness for sides.

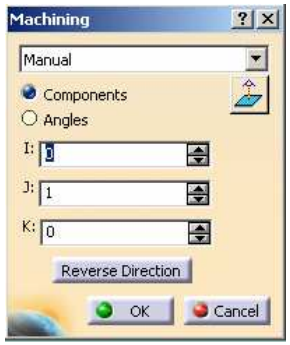


B. Minimum thickness on horizontal areas

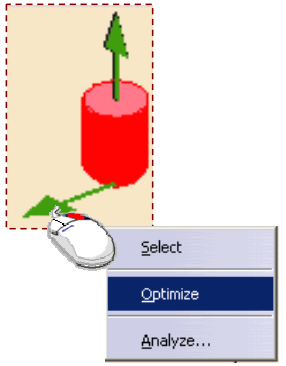




In Back and Forth strategy, machining direction can be set manually using axis definition dialog box. It can be set automatically using optimize option (right mouse button menu).



OR

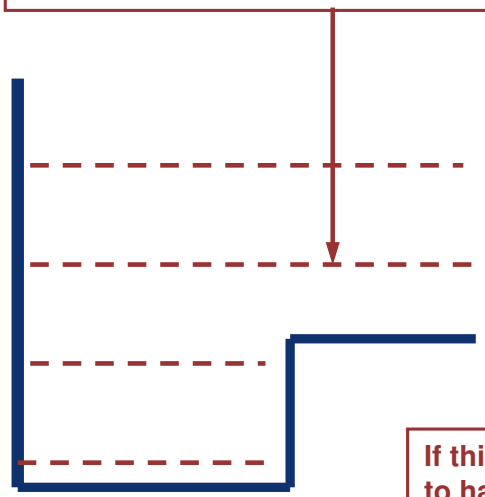


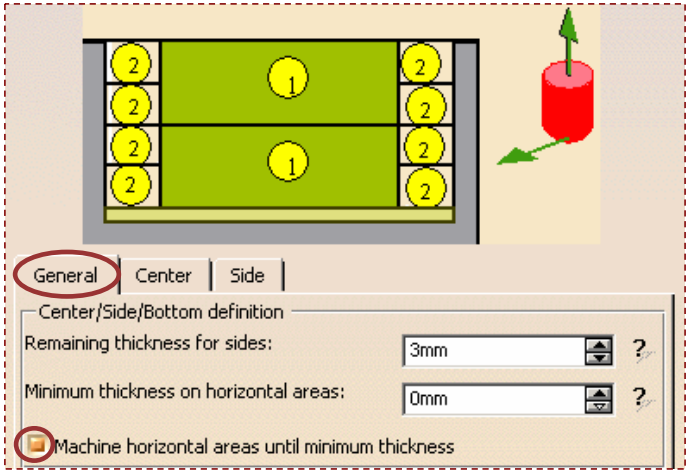
Student Notes:

General Parameters (2/5)

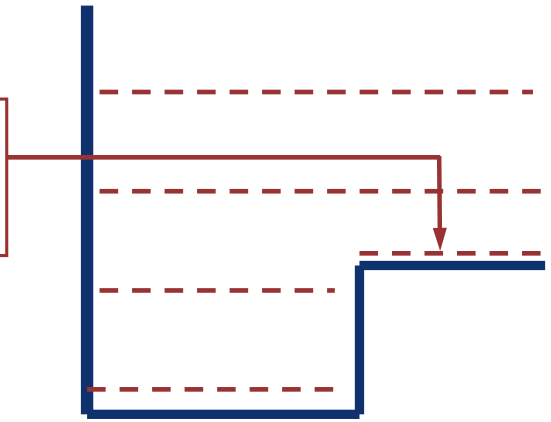
Machine horizontal areas until minimum thickness option:

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.'





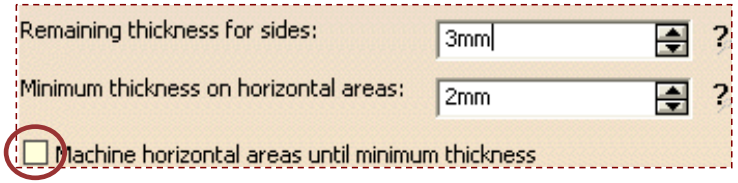
If this option is activated, it will force to have one extra path on this horizontal area to respect minimum thickness.

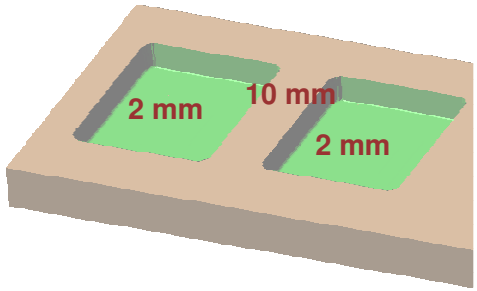


Student Notes:

General Parameters (3/5)

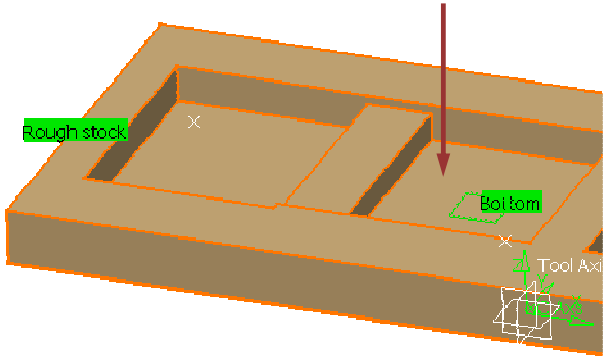
Machine horizontal areas until minimum thickness example:

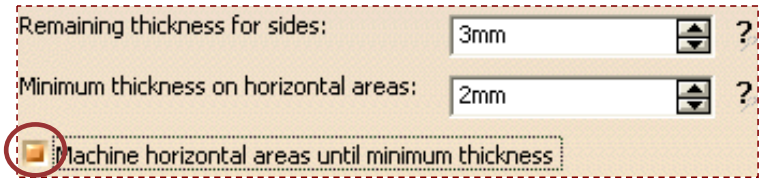


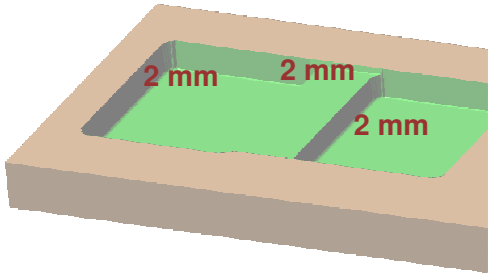


Machine horizontal areas until minimum thickness - Not activated

Bottom plane = bottom of pocket + 2mm offset







Machine horizontal areas until minimum thickness - Activated

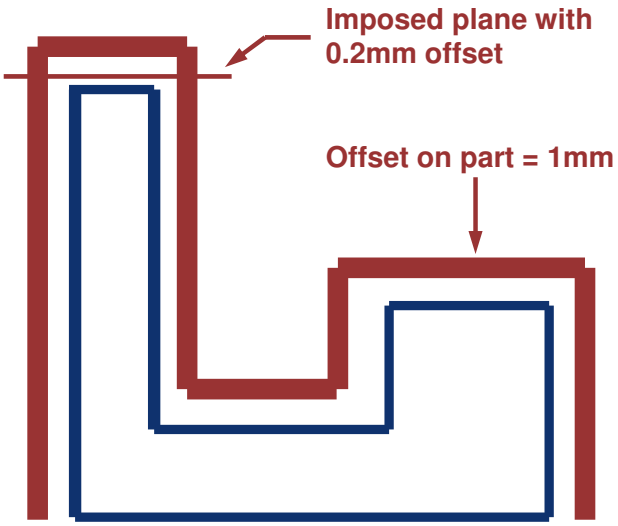
Student Notes:

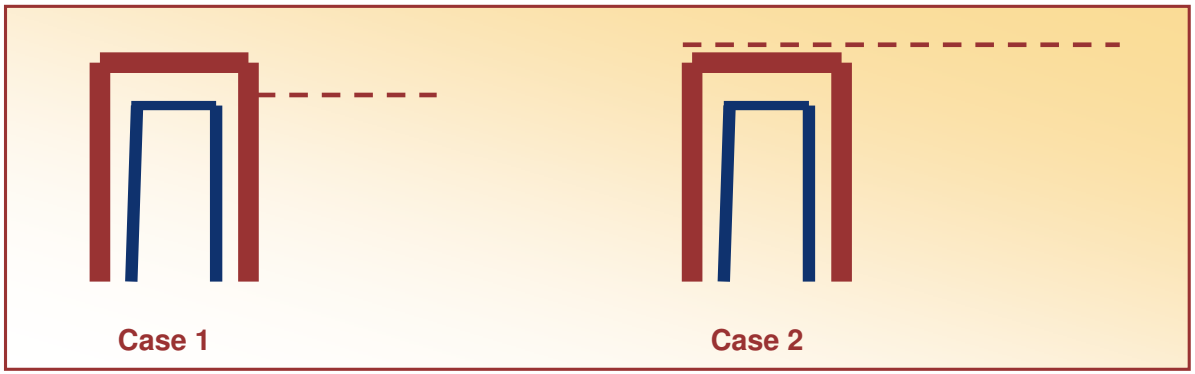
General Parameters (4/5)

Global offset on part vs. imposed plane vs. automatic detection of horizontal area

Global offset on part: This parameter is virtually creating a new part including this offset.

- **Case 1 : If an imposed plane is defined on top of stiffener with 0.2mm offset**
  - **There will not be any tool path on top of stiffener (imposed plane is not visible)**
  - **There will be machining at this height while it is not in collision with the part**
- **Case 2 : If automatic detection is used with the option machining until thickness (still 0.2mm)**
  - **There will be a machining path at 1.2mm height**

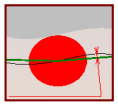


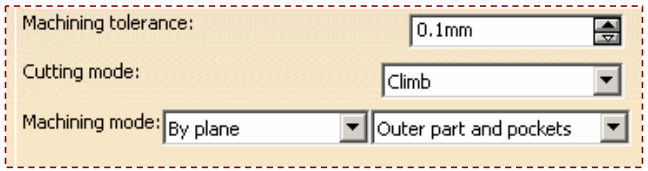


Student Notes:

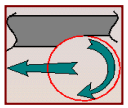
General Parameters (5/5)

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

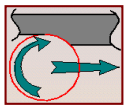




- **Direction of cut definition:**
  - **Climb: The front of the advancing tool cuts into the material first.**



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

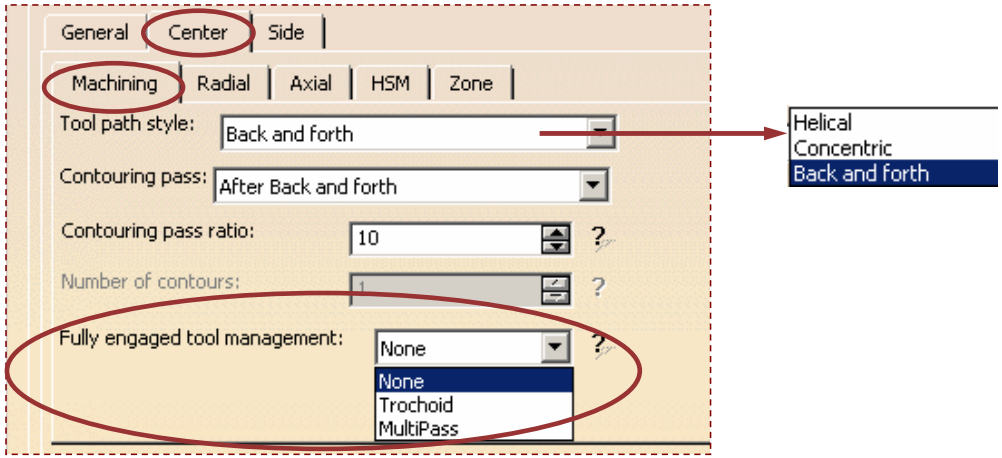


- **Machining mode (refer to outer part and pocket area definition):**
  This option allows to select geometry machining between
  - **Outer part and pocket,**
  - **Pockets only and**
  - **Outer part**

- **Sequencing:**
  - **By plane or**
  - **By area**

Student Notes:

Center Parameters: Machining tab



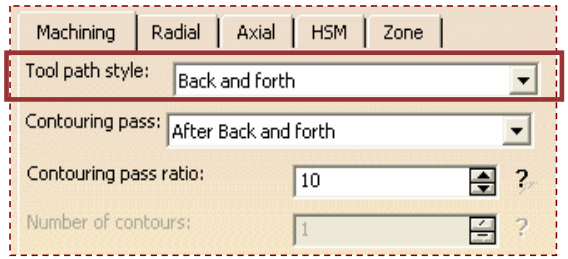
The Fully engaged tool management option is detailed in SMG Roughing. This option is used to optimize the management of tool overload in roughing. Based on automatic detection of full diameter engagement situation, the user has the ability to manage the tool overload by: Feed rate reduction, Extra machining planes or Addition of trochoidal paths.

The main target is the reduction of the machining time and tool life improvement for hard material machining.

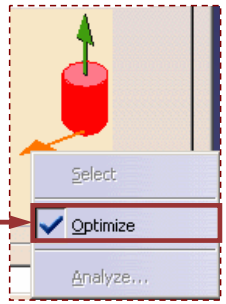
Student Notes:

Center Parameters: Back and Forth

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

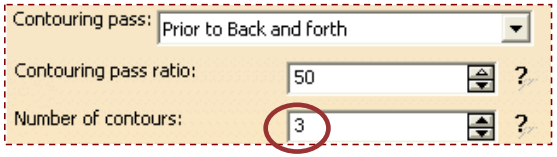


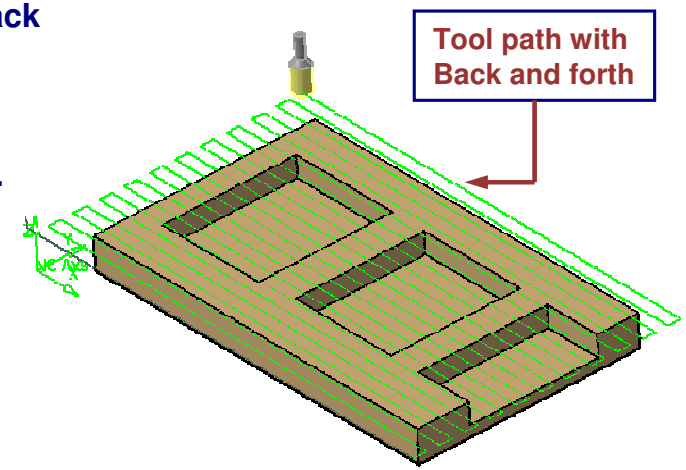
Optimize option let the algorithm choosing direction in order to minimize change of direction in tool path.



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).

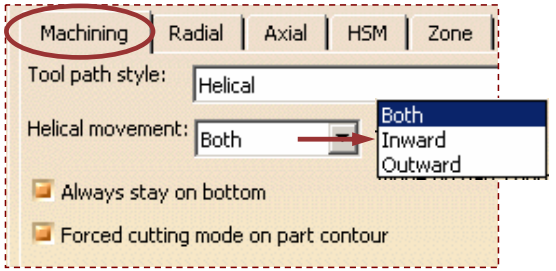




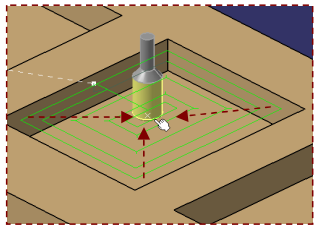
Student Notes:

Center Parameters: Helical (1/4)

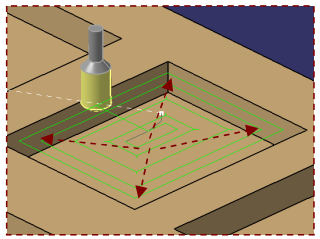
Helical strategy definition:
Tool moves in successive concentric passes from the boundary of the area to machine towards the interior or from the interior to the boundary.

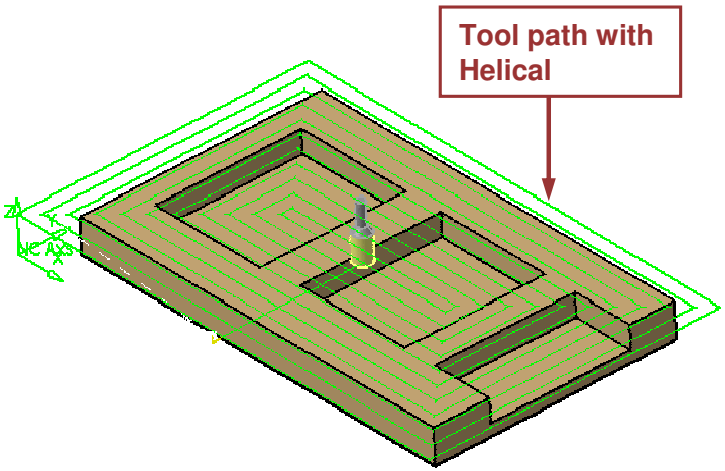


- **Helical Movement:**
  - **Inward:**
    Tools start from a point on zone boundary and follow concentric passes parallel to boundaries towards interior.



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

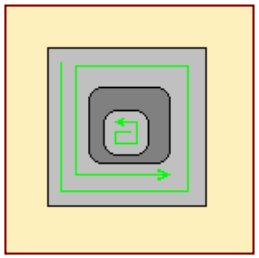


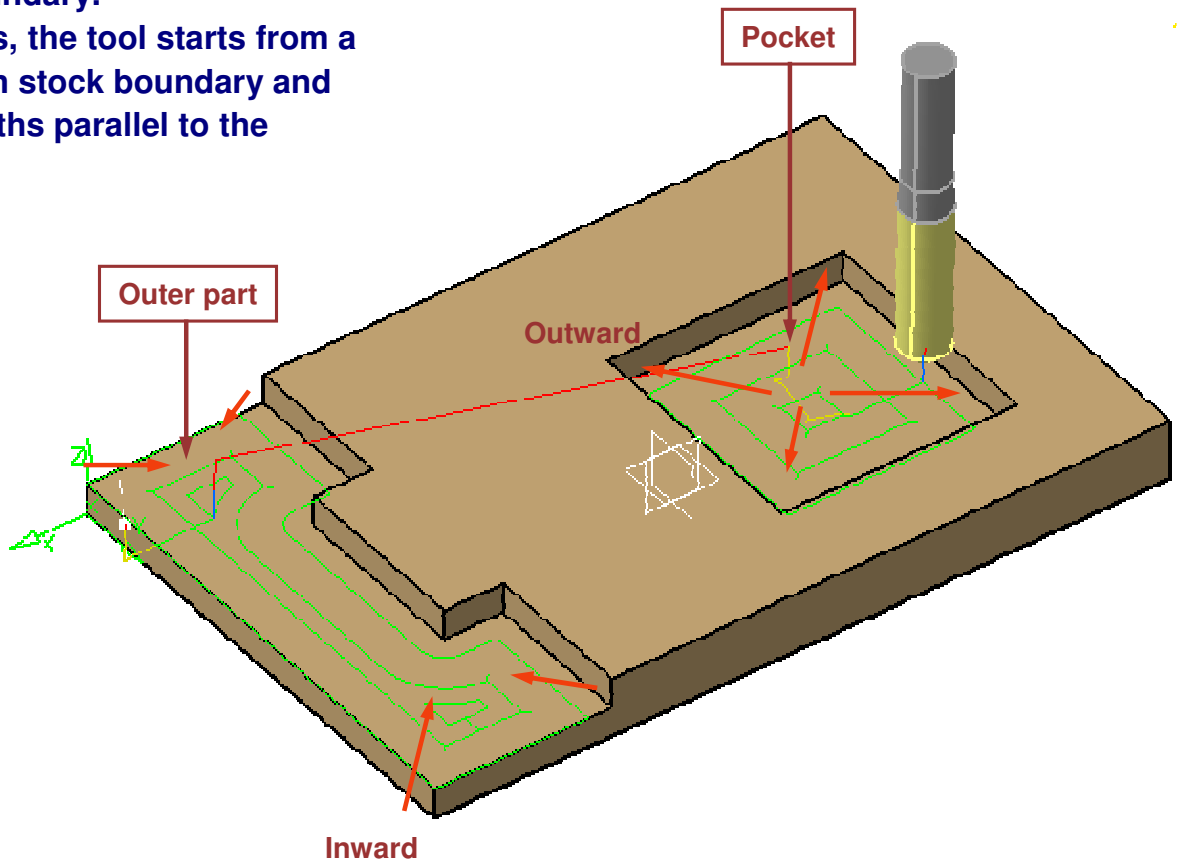


Student Notes:

Center Parameters: Helical (2/4)

- 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.



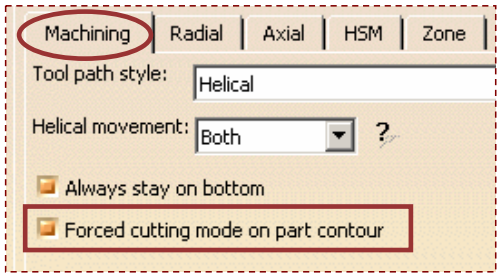


Student Notes:

Center Parameters: Helical (3/4)

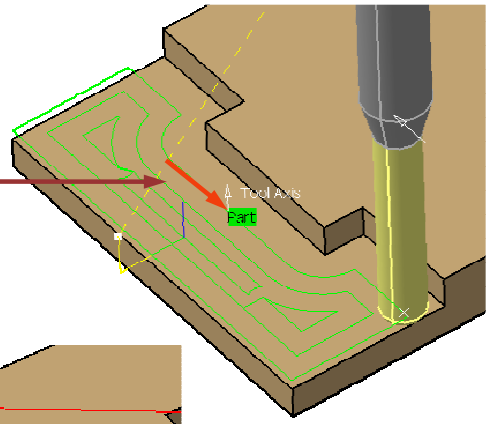
Forced cutting mode on part contour:

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:

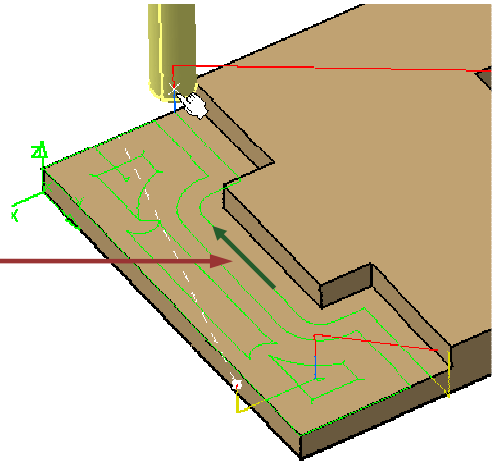
- **Inward**
- **contouring pass is not respecting climb**



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**



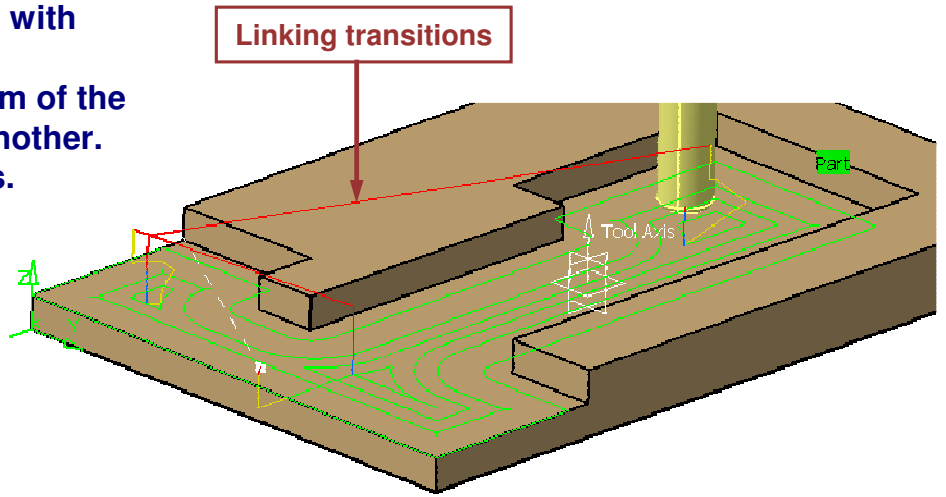
Student Notes:

Center Parameters: Helical (4/4)

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.

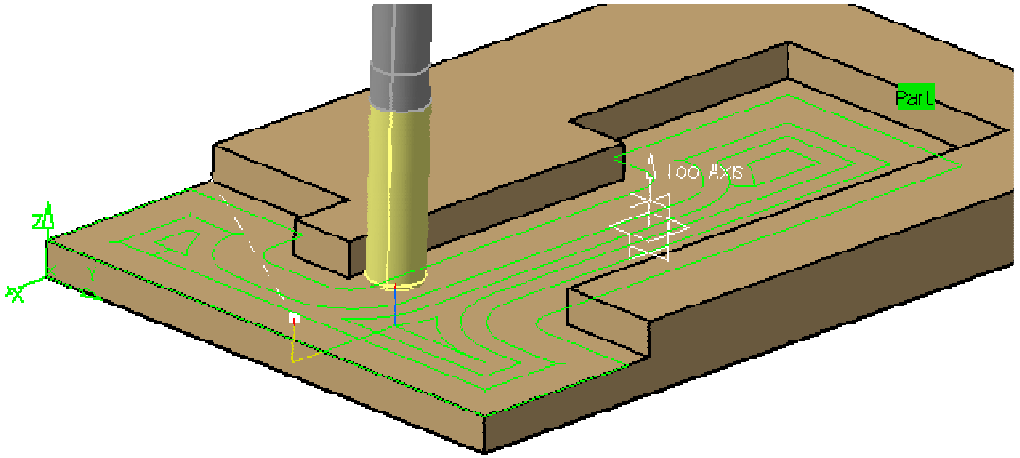
Option OFF:

- Approach macro
- Retract macro
- 2 linking movements



Option ON:

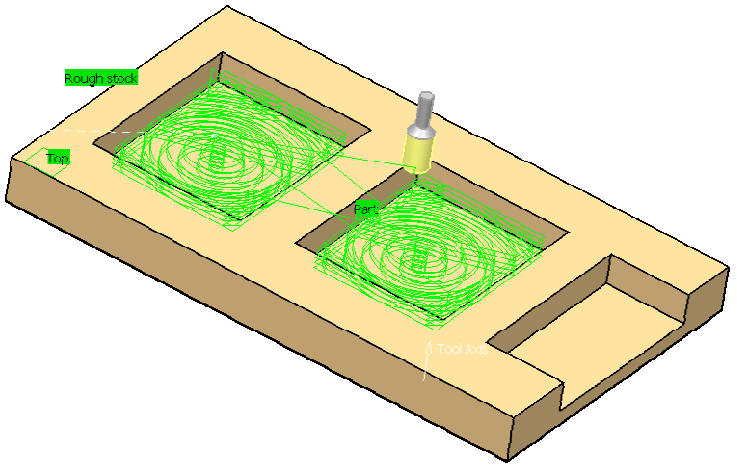
- Approach macro
- Retract macro
- No linking movements



Center Parameters: Concentric

Concentric strategy definition:

- **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.**



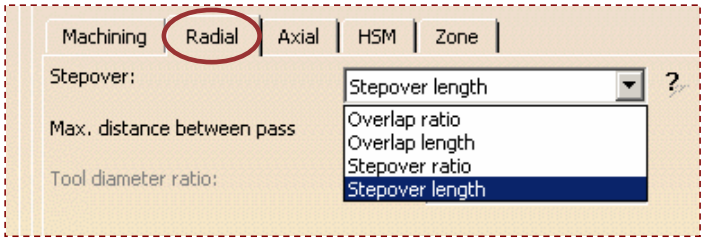
Student Notes:

Student Notes:

Center Parameters: Radial tab

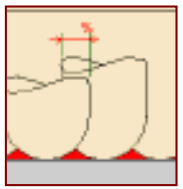
There are four different ways to define distance between passes:

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

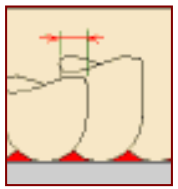


Overlapping

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

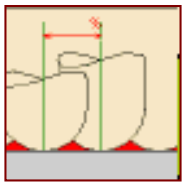


Overlap length:
It is the distance between two passes with respect to a tool diameter ratio recovery.

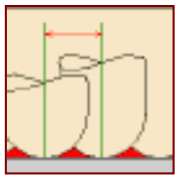


Stepover

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



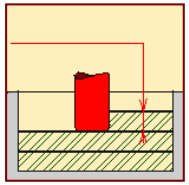
Stepover length:
It is the maximum distance between two passes.

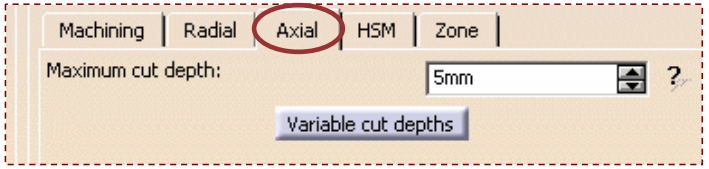


Student Notes:

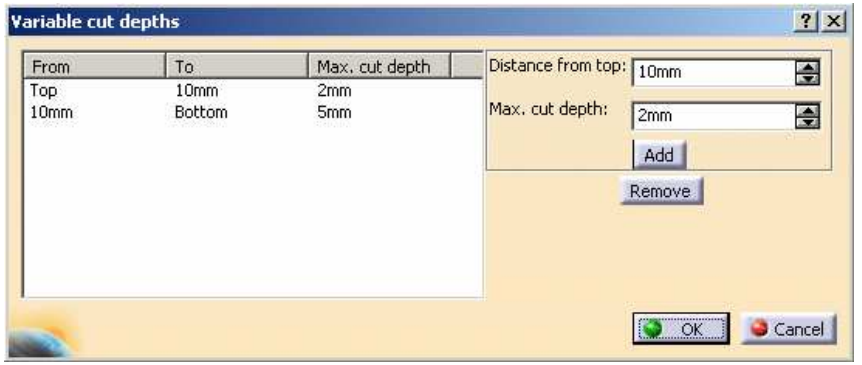
Center Parameters: Axial tab

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.

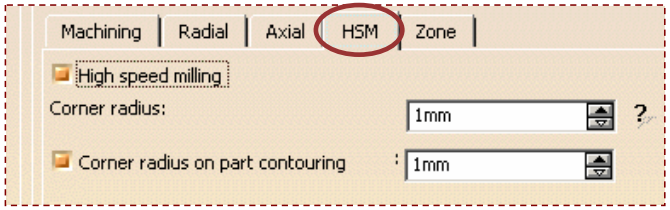


Student Notes:

Center Parameters: 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.

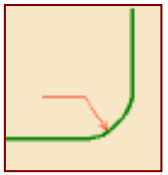


It is possible to have a different cornerization on part contouring (most of the time a smaller one to reduce rework).

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.

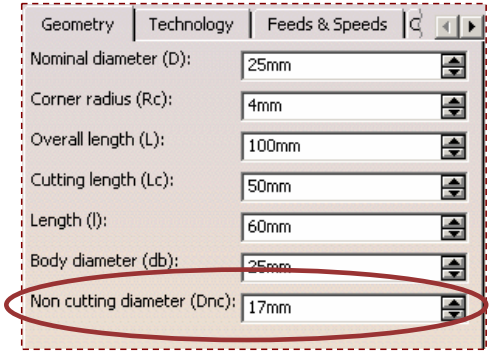
Student Notes:

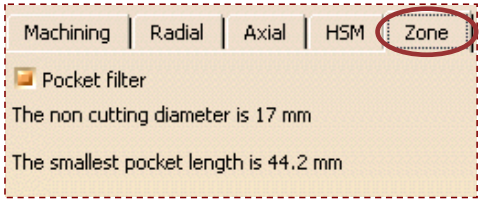
Center Parameters: Zone tab

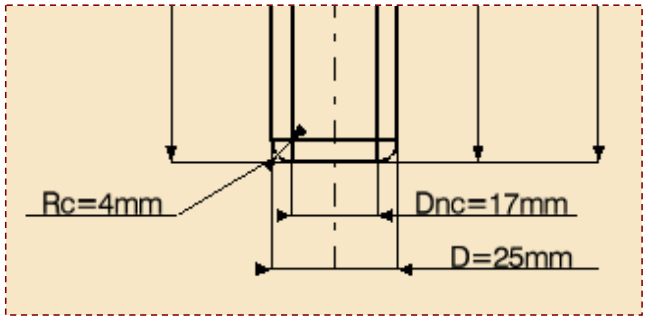
Zone definition:

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

To be activated you 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)

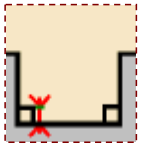
There will not be machining path in pockets where tool can't plunge without respecting maximum plunge angle.

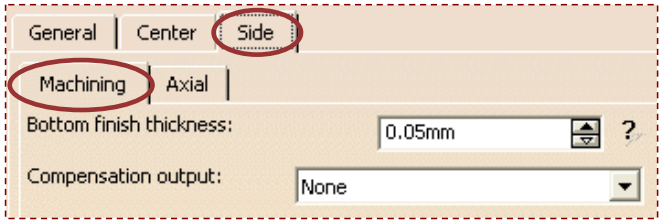
Student Notes:

Side Parameters

Machining tab:

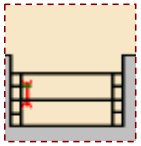
Bottom finish thickness: Define the thickness value left on bottom of part during last level of side finish tool path





Compensation output: Automatic insertion of CUTCOM instruction in tool path to manage tool compensation

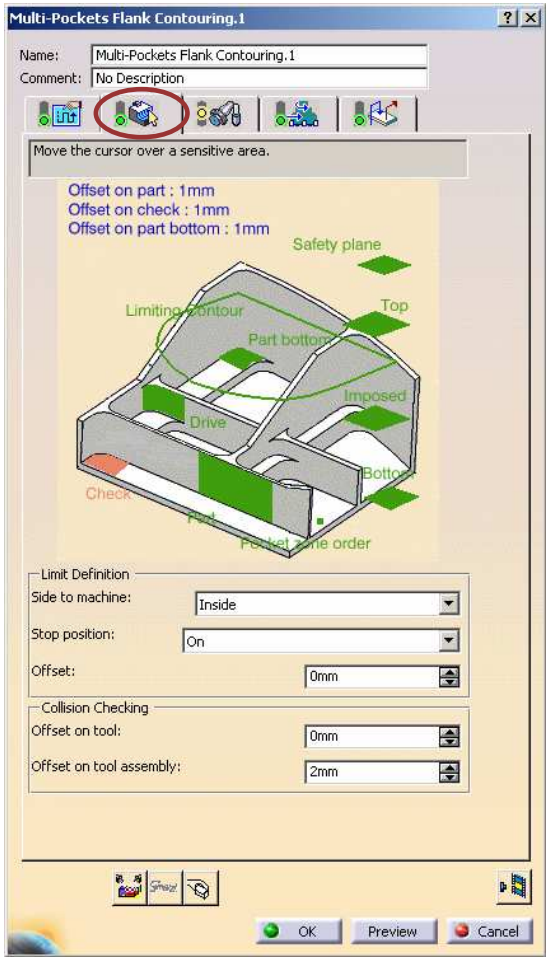
Axial tab:
Maximum cut depth



Student Notes:

Multi-Pockets Flank Contouring: Geometry

You will see the options in the Geometry tab of Multi-Pockets Flank Contouring .



Student Notes:

Presentation (1/2)

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

- **A : Part**

  Multi-pocket flank contouring operation operation will machine the part with multiple pockets. Offset can be applied on part.

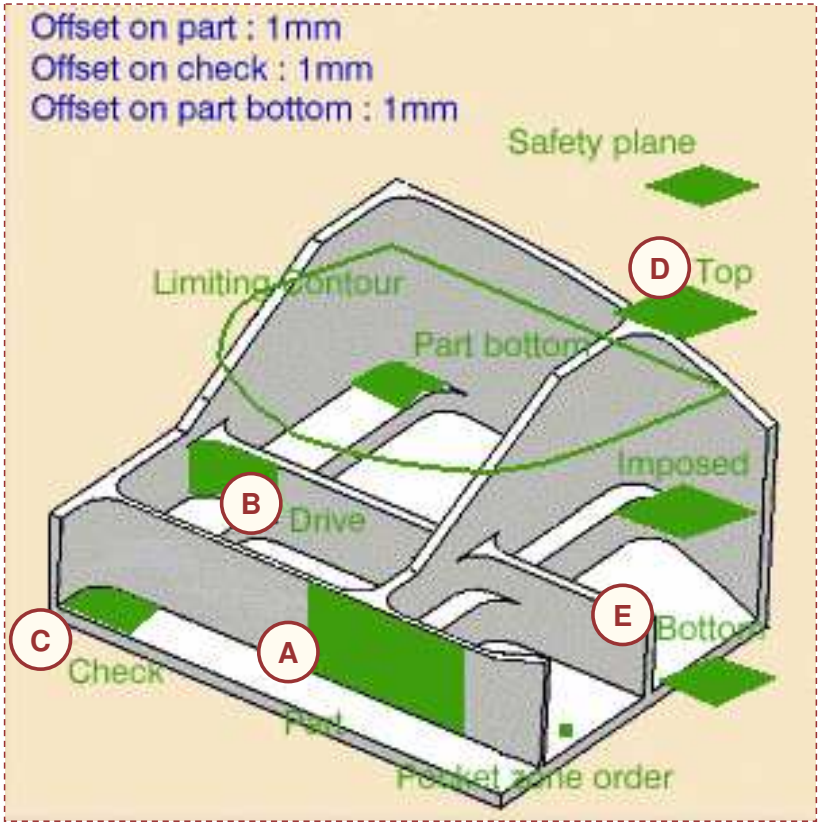
- **B : Drive**
  Element that determines the drive surfaces to be followed by the flank of the tool.

- **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.

  Offset can be applied on part bottom.



Student Notes:

Presentation (2/2)

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

- **F : Imposed planes**

  Force cutter to machine in this plane (global offset can be applied on each group)

- **G : Safety plane**

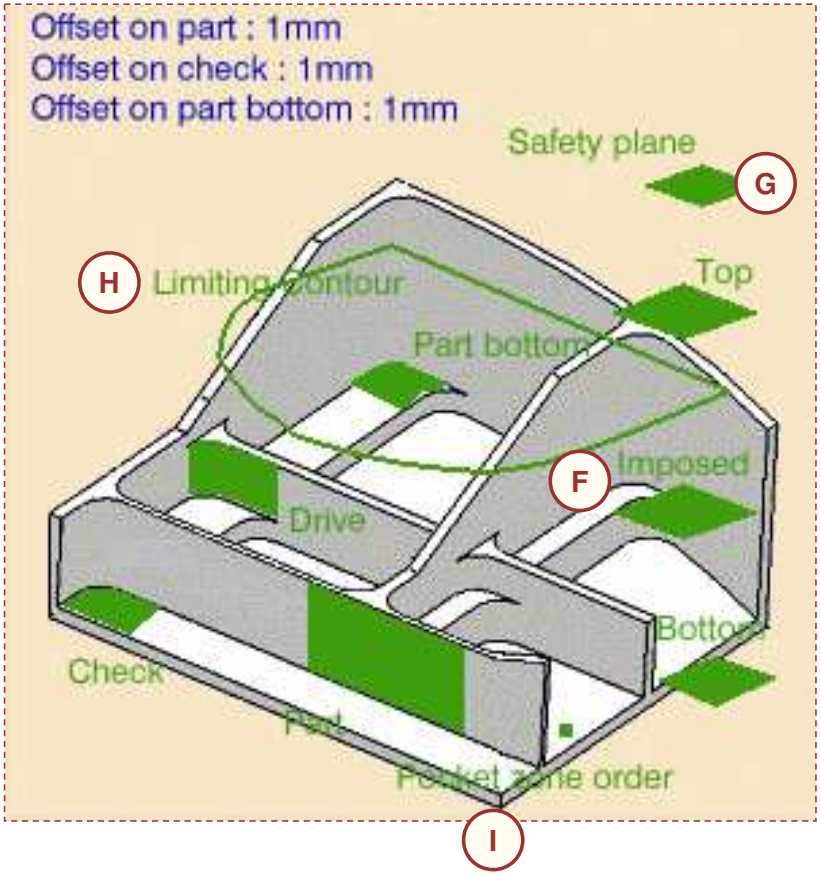
  The plane that the tool will rise to at the end of the tool path in order to avoid collisions with the part.

- **H : Limiting contour**

  Re-limit machining area after stock and part definition. It is a 2D limitation along the view direction

- **I : Pocket zone order**

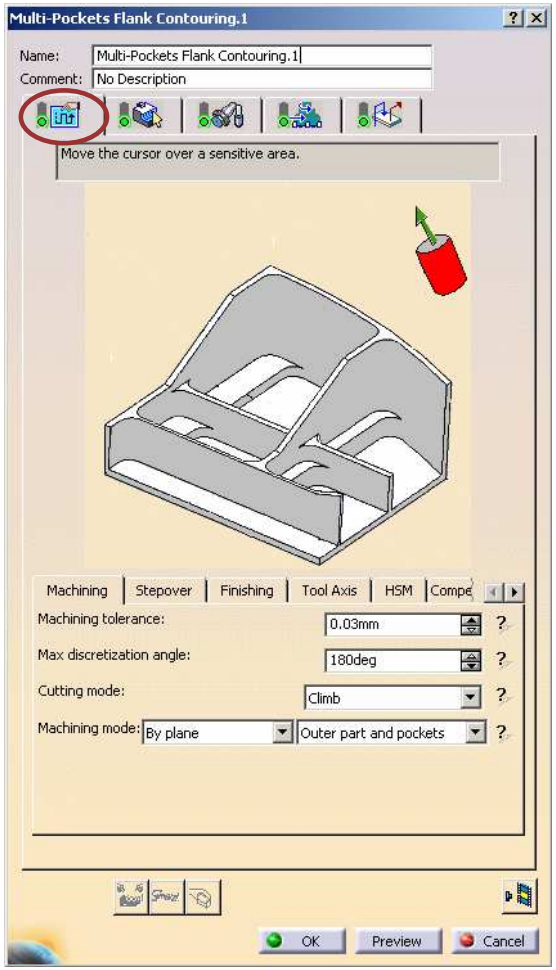
  Define pocket machining order



Student Notes:

Multi-Pockets Flank Contouring: Strategy

You will learn the options in the Strategy tab of Multi-Pockets Flank Contouring.

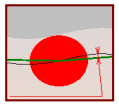


Strategy (1/3)

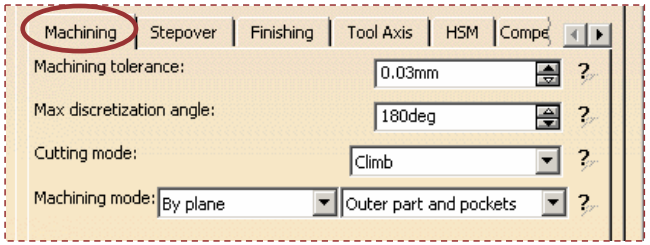
Student Notes:

Machining

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



- **Max discretization angle: Maximum angle between two consecutive points that the machine is able to achieve.**

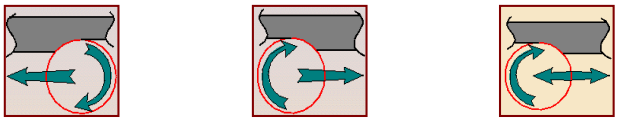


- **Machining mode :**

  By plane: The whole part is machined plane by plane

  By area: The whole part is machined area by area

- **Cutting mode:**

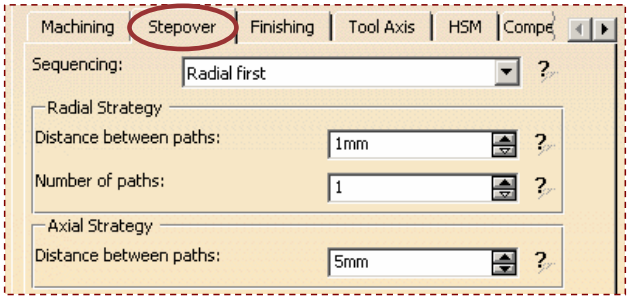


Climb **Conventional** **Either**

Select geometry machining between Outer part and pockets, Pockets only and Outer part

Stepover:

It defines Radial and Axial parameters.

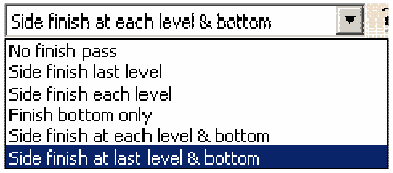


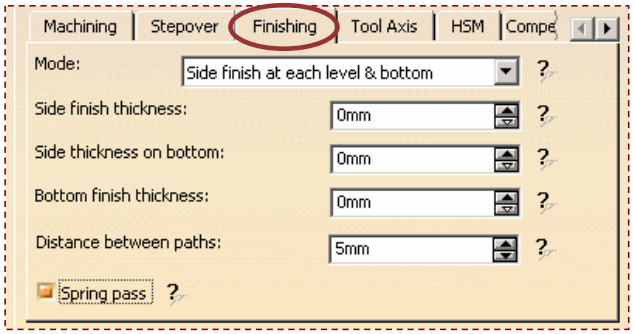
Student Notes:

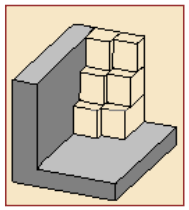
Strategy (2/3)

Finishing

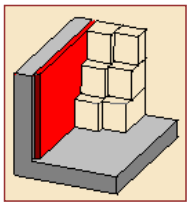
Modes:



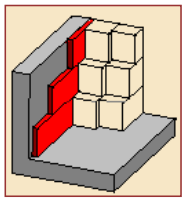




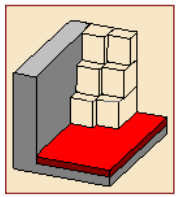
No finish pass: Only one Side finish is added by side finish level



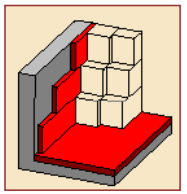
Side finish last level: Only one Side finish is added on the last level



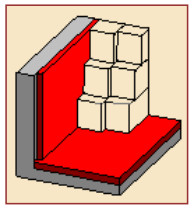
Side finish each level: One Side finish pass is added by the finishing level



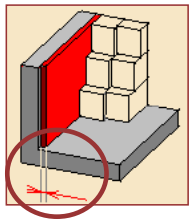
Finish bottom only: The last passes where the tool is in contact with the bottom detected



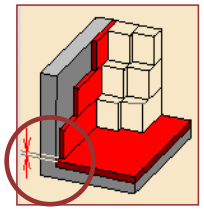
Side finish at each level & bottom: Addition of Side finish at each level and finish bottom



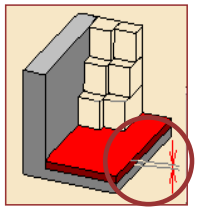
Side finish at last level & bottom: Addition of Side finish at last level and finish bottom



Side finish thickness



Side thickness on bottom



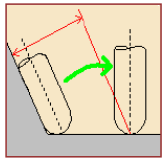
Bottom finish thickness

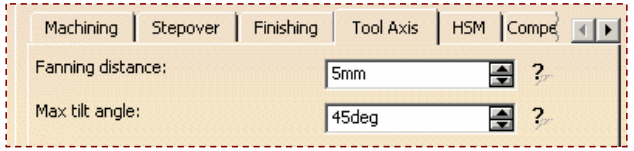
Student Notes:

Strategy (3/3)

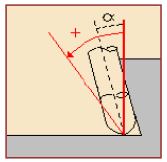
Tool Axis

- **Fanning distance: The distance at the beginning and the end of the motion where fanning takes place.**



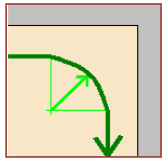


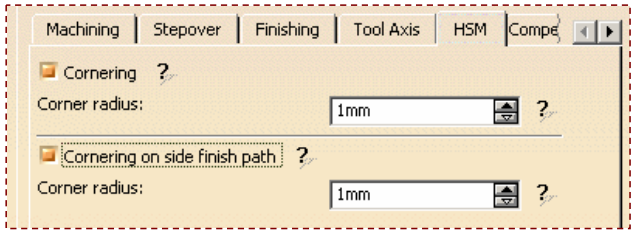
- **Max tilt angle: The max angle at which the tool axis can tilt.**



HSM

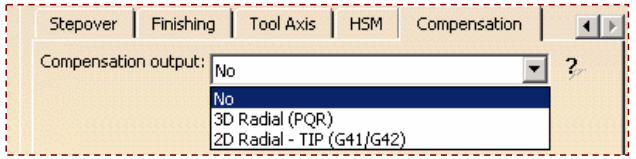
- **Cornering: defines the corner radius.**
- **Cornering on side finish path: The corner on side finish path radius value.**

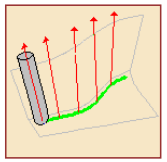




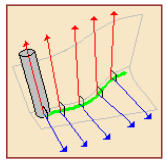
Compensation

- **Compensation output:**

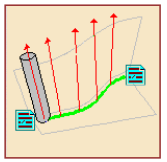




No



3D Radial (PQR)



2D Radial- TIP (G41/G42)

Student Notes:

Managing Offsets

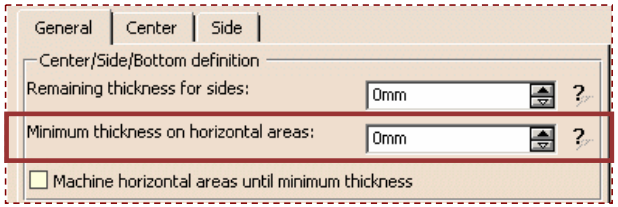
You will learn the offset management in detail.

Student Notes:

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

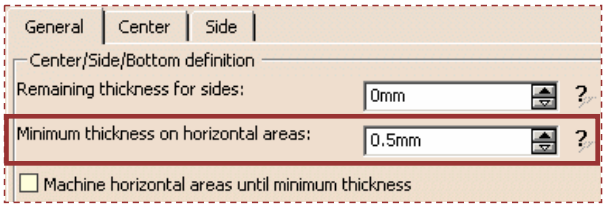
H : depth to remove
D : max depth of cut
N : number of level

Student Notes:

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

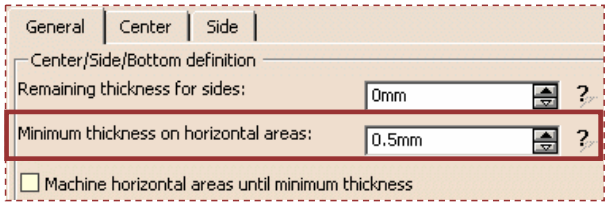
H : depth to remove
D : max depth of cut
N : number of level

Student Notes:

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

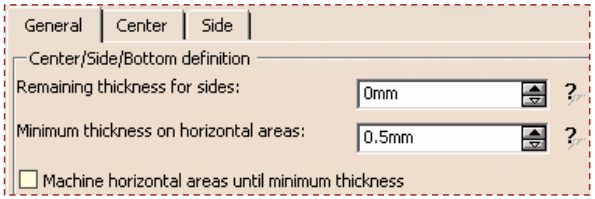
H : depth to remove
D : max depth of cut
N : number of level

Student Notes:

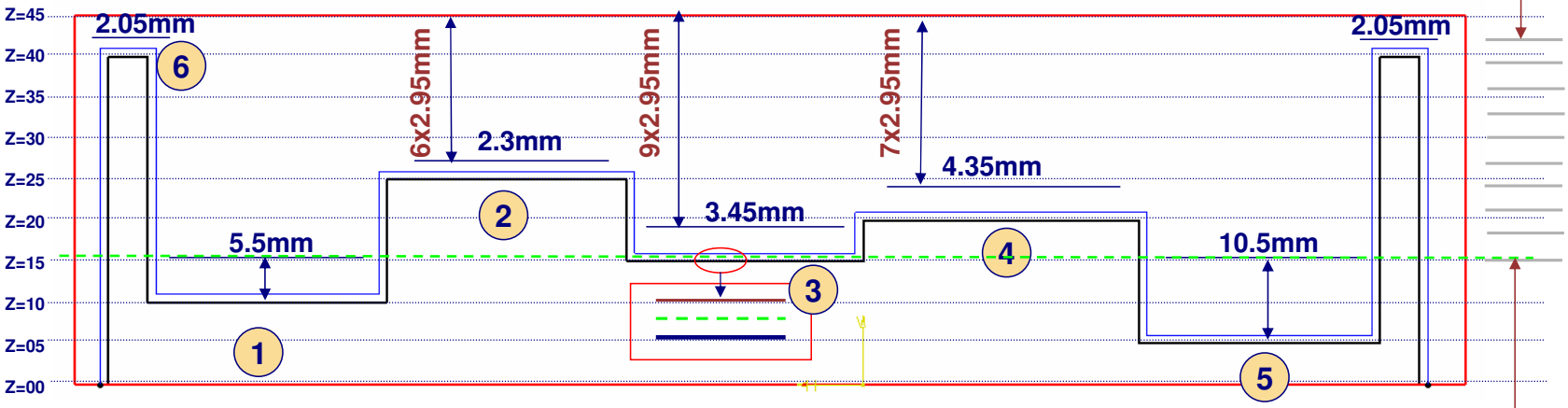
Case 4: Bottom Plane

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

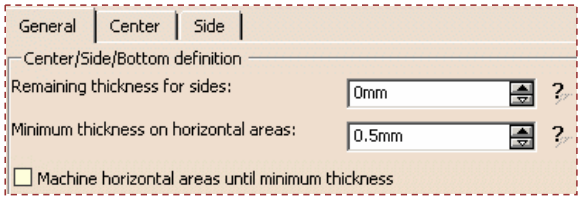
H (top-bottom): depth to remove from top of the stock to bottom plane
N: number of level
The bottom path is done only in zones 1 & 5.

Student Notes:

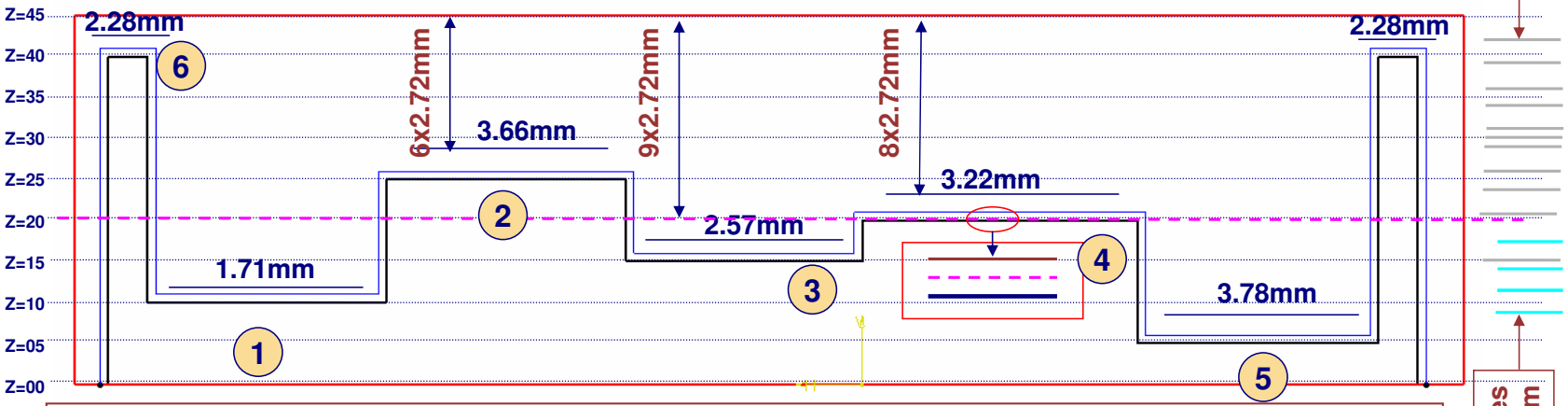
Case 5: Imposed Plane

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)



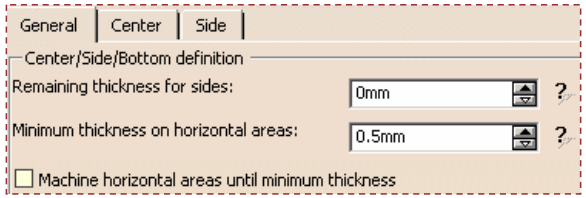
1. Recomputed depth to have regular depth of cut between imposed planes
until imposed plane: Recompute depth: H (top-imposed plane)/N closest than max depth of cut = 2.72 mm
after imposed plane: Recompute depth: H (imposed plane-last plane)/N closest than max depth of cut = 2.93 mm
2. Compute of the 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:

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

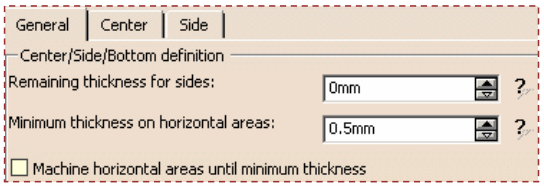
The zone 6 is not machined because there are upper top plane.

Student Notes:

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:

To Sum Up

In this course you have seen:

- **Necessary geometrical elements to define Power Machining and Multi-Pockets Flank Contouring operations**
  - **Part (can be composed of different elements)**
  - **Stock**
  - **Planes (top, bottom, imposed)**
- **General parameters**
- **Center parameters**
  - **Machining strategies of Power Machining**
  - **Helical, Back and Forth, both with HSM option**
  - **Radial and Axial strategies**
- **Side parameters**
- **Added Exercise**