Prismatic Machining

Preface

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

It offers an easy-to-use and easy-to-learn graphic interface that makes it suitable for shop floor-oriented use. Moreover, its leading edge technologies together with a tight integration with Version 5 design methodologies and DELMIA's digital manufacturing environment, fully satisfy the requirements of office programming. Prismatic Machining is a unique solution that reconciliates office and shop floor activities.

It is integrated to a Post Processor Access execution engine, allowing the product to cover the whole manufacturing process from tool trajectory (APT source) to NC code.

This product is particularly adapted for tooling and simple machined parts, and is also an ideal complement to other manufacturing applications.

Prismatic Machining offers the following main functions:

- 2.5 axis milling and drilling capabilities
- Management of tools and tool catalogs
- Flexible management of the manufacturing program with intuitive and easy-to-learn user interface based on graphic dialog boxes
- Tight interaction between tool path definition, verification and generation
- Seamless NC data generation thanks to an integrated Post Processor Access solution
- Automatic shop floor documentation in HTML format
- High associative level of the manufacturing program ensures productive design change management thanks to the integration with Version 5 modeling capabilities
- Based on the Process Product Resources (PPR) model, the manufacturing applications are integrated with Digital Process for Manufacturing (DPM).

What's New?

Milling Operations

New: Prismatic Rework capability for Pocketing and Profile Contouring operations.

Enhanced: A sequence of tool motions can now be defined using part, drive and check elements in a Point to Point operation.

Enhanced: Multi-contours can now be machined in a Profile Contouring operation.

Enhanced: Contour and island detection from part bottom selection.

Axial Machining Operations

Enhanced: Additional types of macro path on axial operations.

Enhanced: Capability to avoid collisions with Fixtures and Check elements.

General Functions

Enhanced: This product benefits from the enhancements brought to the NC Manufacturing Infrastructure.

Getting Started

Before getting into the detailed instructions for using Prismatic Machining, this tutorial is intended to give you a feel of what you can accomplish with the product.

It provides the following step-by-step scenario that shows you how to use some of the key functionalities.

Entering the Workbench

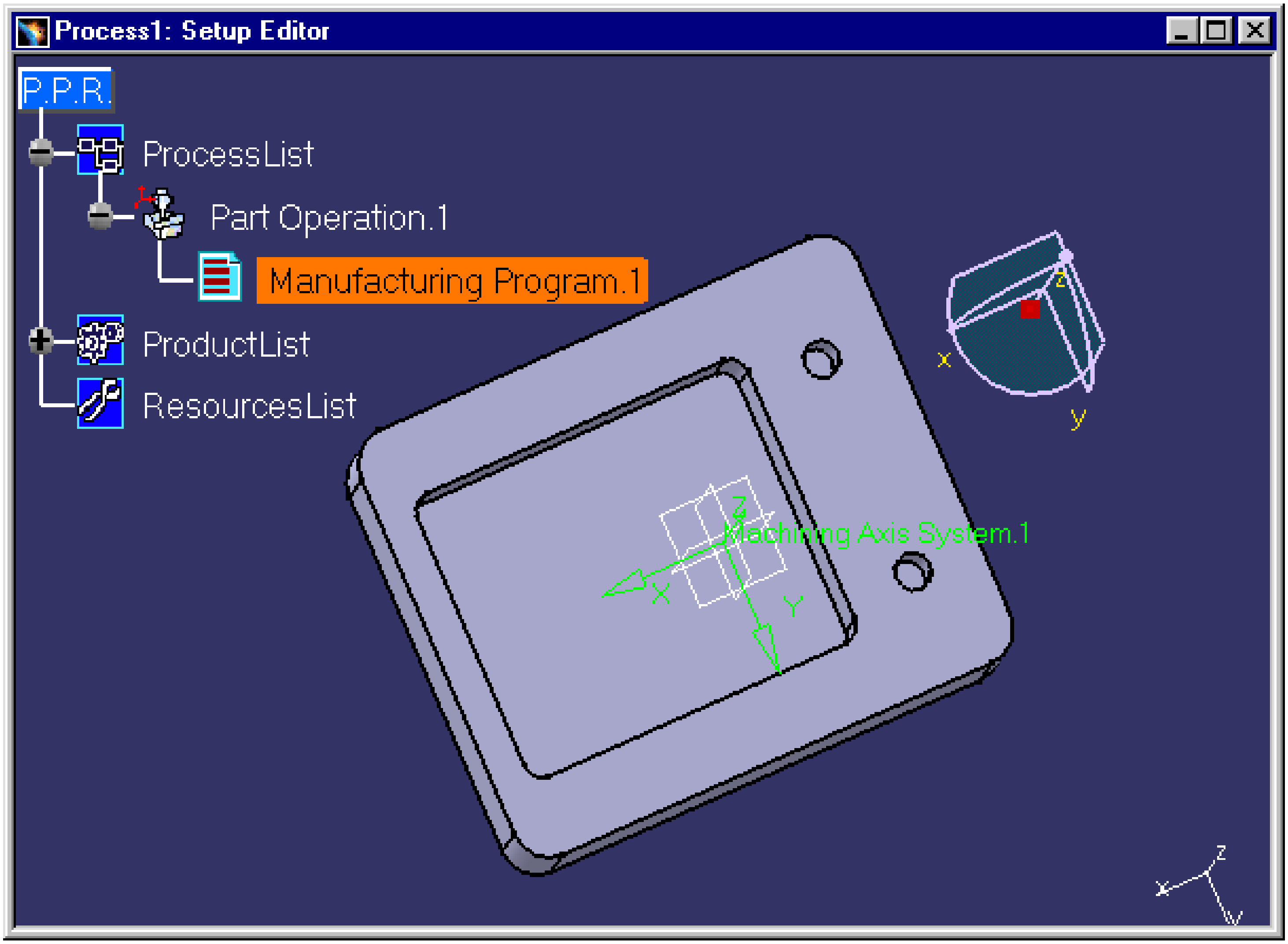
This first task shows you how to open a part and enter the Prismatic Machining workbench.

1. Select File -> Open then select the GettingStartedPrismaticMachining.CATPart document.

2. Select NC Manufacturing > Prismatic Machining from the Start menu.

The Prismatic Machining workbench appears.

The part is displayed in the Setup Editor window along with the manufacturing specification tree.



3. Select Manufacturing Program.1 in the tree to make it the current entity.

To insert program entities such as machining operations, tools and auxiliary commands you can either:

- make the program current before clicking the *insert program entity* command
- click the *insert program entity* command then make the program current.

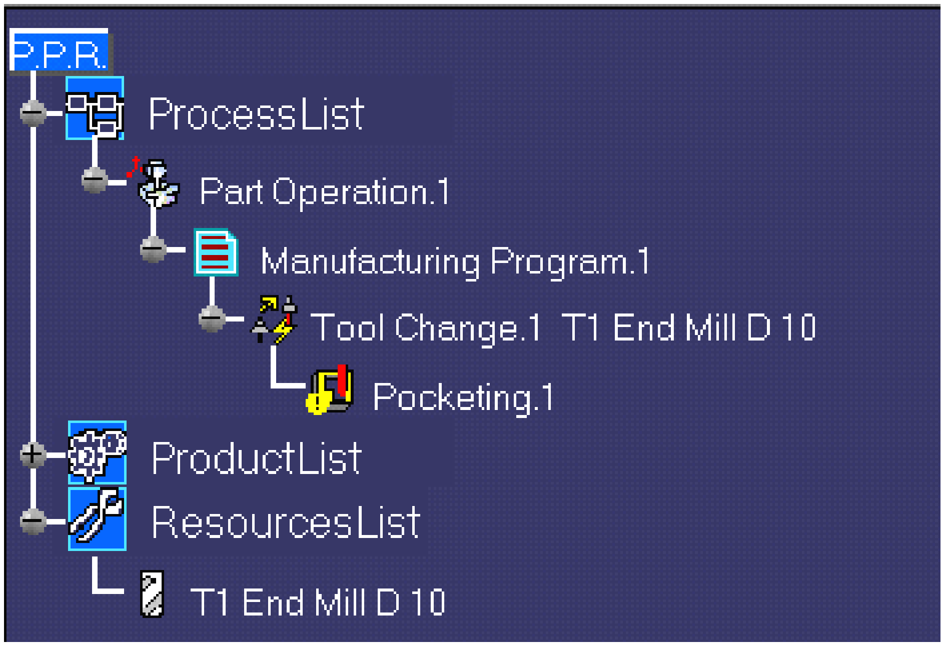
Create a Pocketing Operation

This task shows you how to insert a pocketing operation in the program.

As this operation will use the default tool and options proposed by the program, you just need to specify the geometry to be machined.

1. Select the Pocketing icon .

   A Pocketing.1 entity along with a default tool is added to the program.

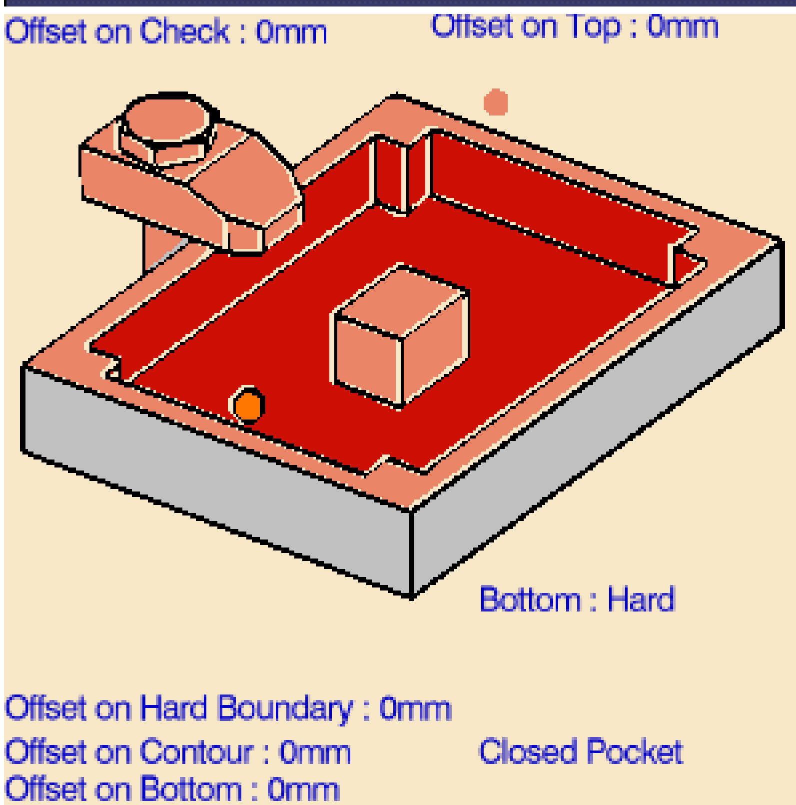


   The Pocketing dialog box appears directly at the Geometry tab page .

   The red status light on the tab indicates that you must select the pocket geometry in order to create the operation.

   The Geometry page includes an icon representing a simple pocket.

   There are several sensitive areas and texts in the icon to help you specify the pocket geometry.



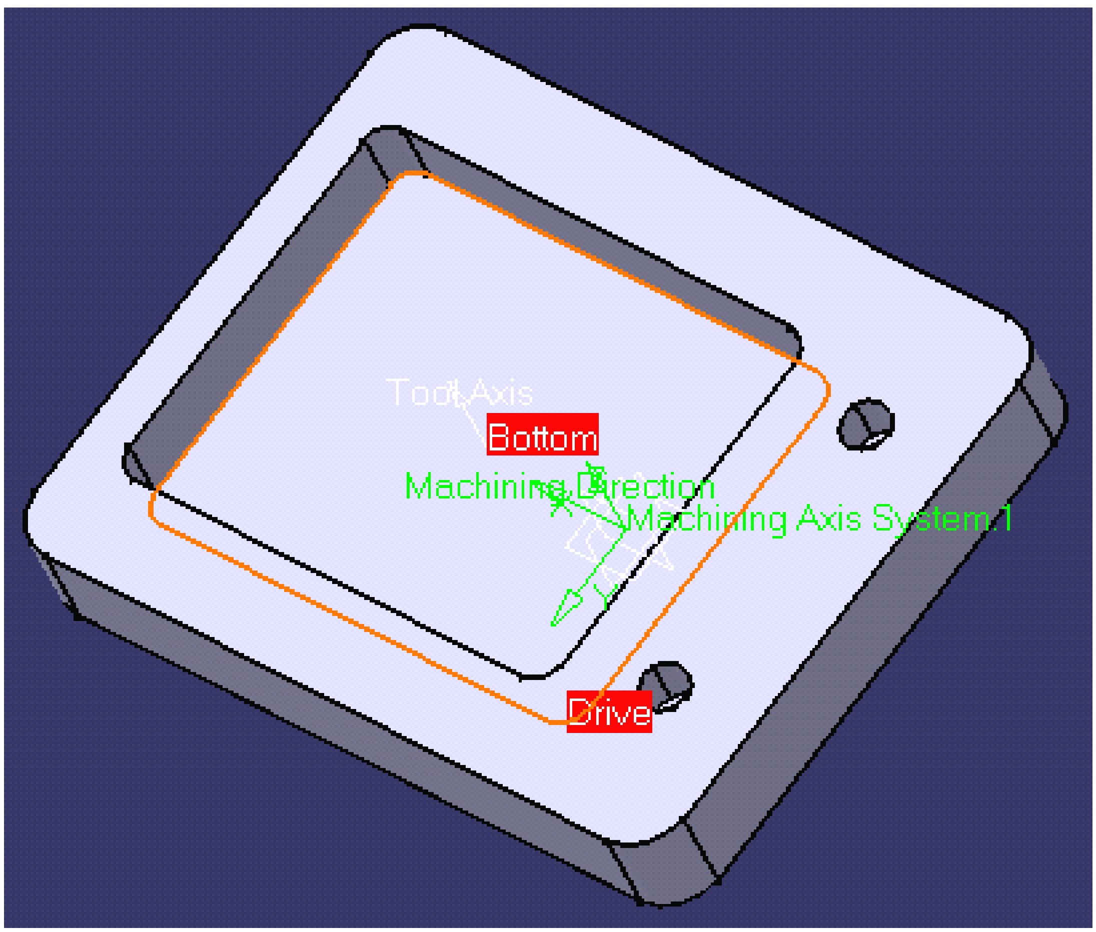
Sensitive areas that are colored red indicate required geometry.

2. Right click the red Bottom in the icon and select **Contour Detection** from the contextual menu.

   Click the red Bottom area.

   The dialog box is reduced allowing you to select the corresponding part geometry.

3. Select the bottom of the pocket.

   The boundary of the selected pocket bottom is automatically proposed as drive element for the operation thanks to the Contour Detection setting

   The dialog box reappears.

   The bottom and sides of the pocket in the icon are now colored green, indicating that the corresponding geometry is defined for the operation. The tab status is now green .

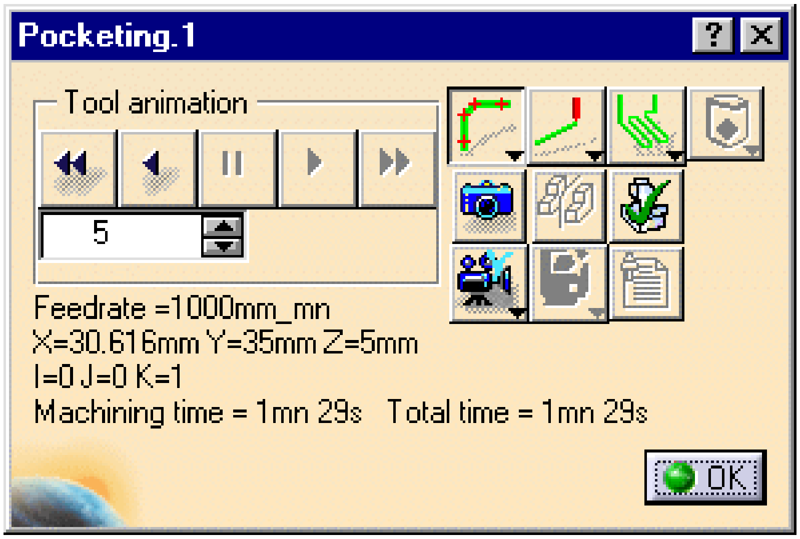
4. Click OK to create the operation.

Replay the Tool Path

This task shows you how to replay the tool path of the pocketing operation.

1. Select the pocketing operation in the tree then select the Replay Tool Path icon.

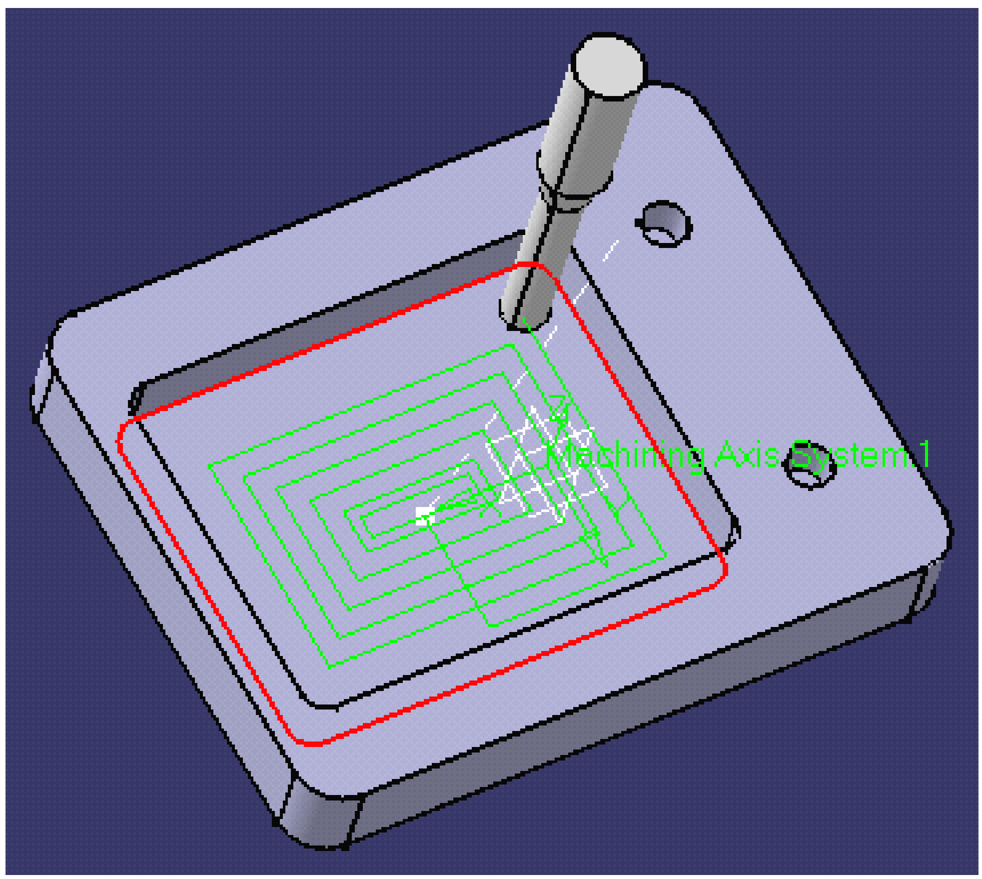
   The tool path is computed and the Replay dialog box appears.



2. Choose the Point by Point replay mode by means of the drop down icon.

   You can set the number of computed points to be replayed at each step of the verification by means of the spinner.

3. Click the button to position the tool at the start point of the operation.



4. Click the button to start the replay and continue to click that button to move the tool along the computed trajectory.

5. Click OK to quit the replay mode.

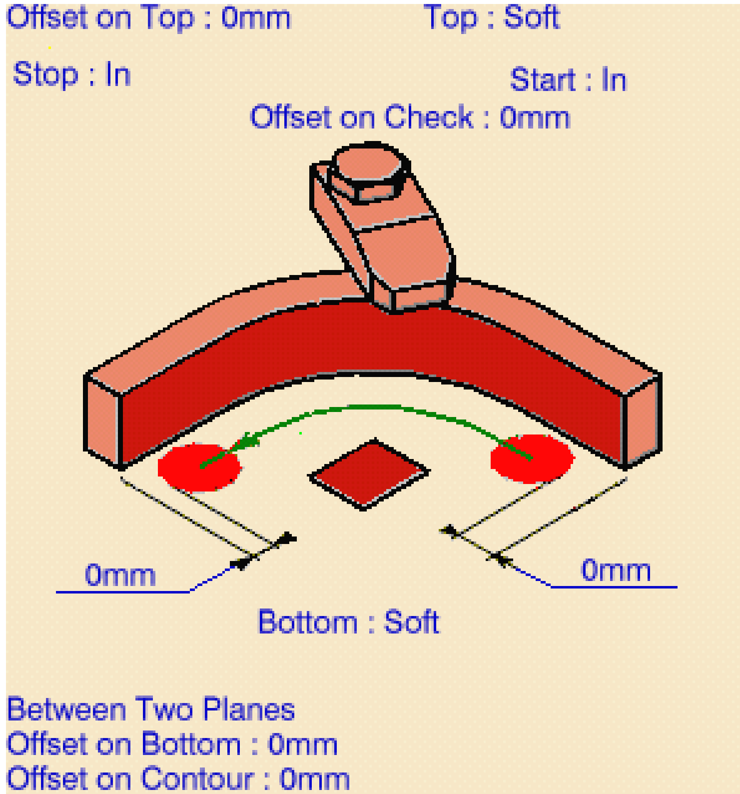
Create a Profile Contouring Operation

This task shows you how to insert a profile contouring operation in the program.

Make sure that the pocketing operation is the current entity in the program.

1. Select the Profile Contouring icon.

   The Profile Contouring dialog box appears directly at the Geometry page.



2. Click the Bottom: Hard text in the sensitive icon to switch the type of bottom to Soft.
3. Click the Bottom plane then select the corresponding part geometry (that is, the underside of the part).

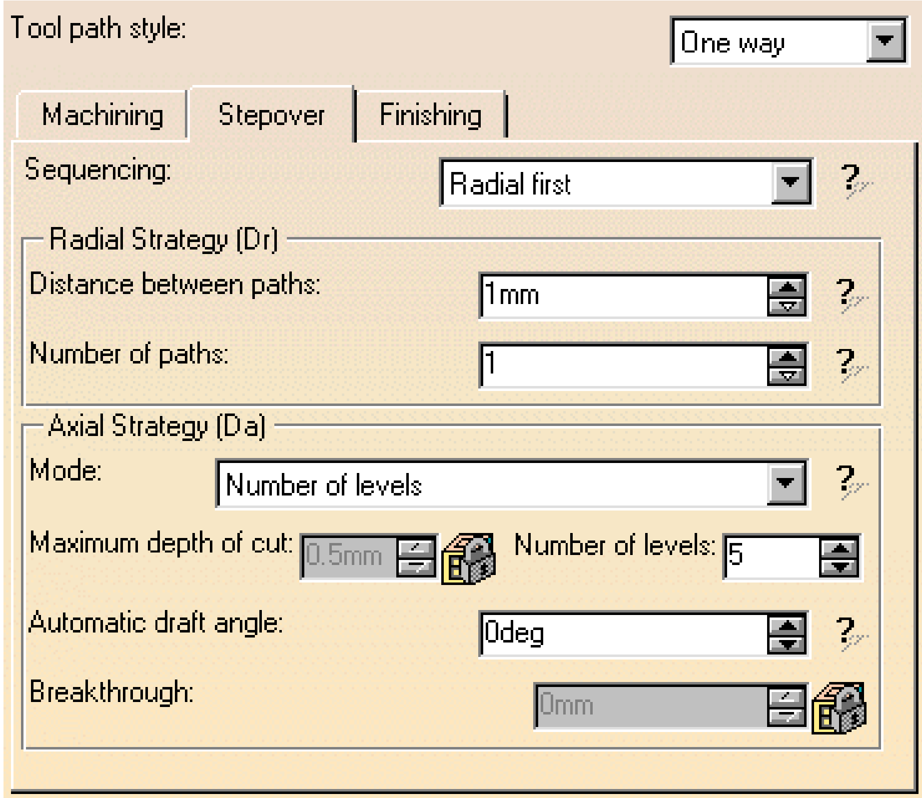
   The closed external contour of the bottom is proposed as Guide element for the operation.

Make sure that the arrow on the Guide element is pointing away from the part.

4. Click the Top plane in the icon, then select the corresponding part geometry.
5. Double click Offset on Contour in the icon.

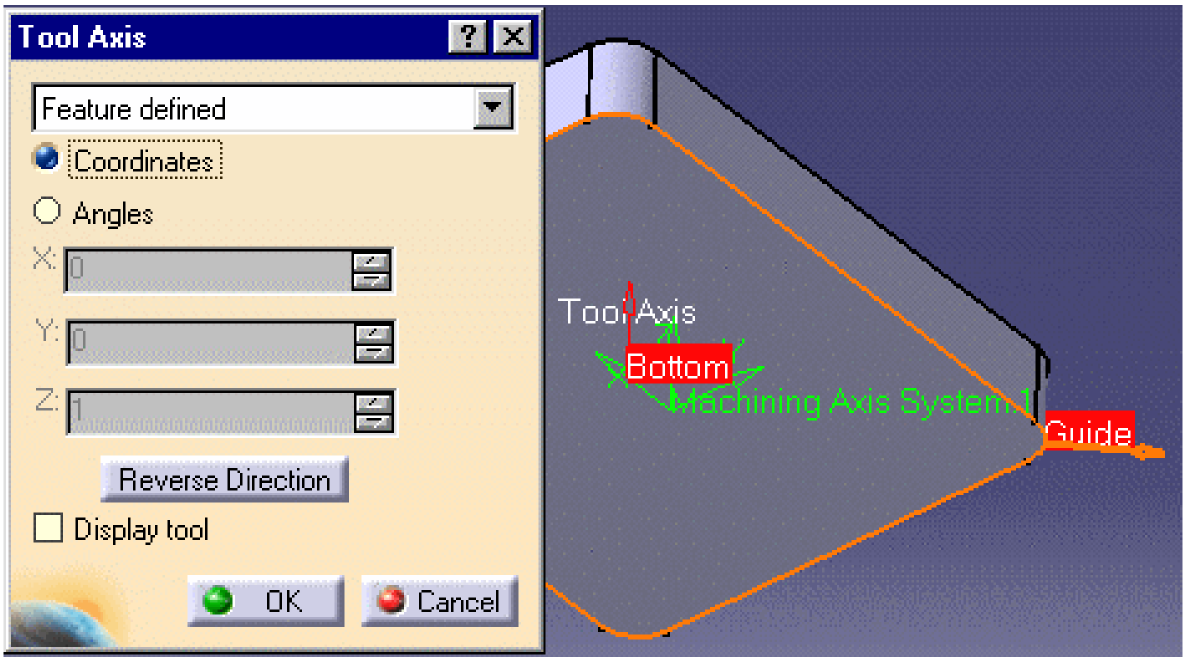
   Set this value to 1mm in the Edit Parameter dialog box and click OK.

6. Select the Strategy tab page and set the parameters as shown.



7. If needed, you can change the tool axis orientation. Just click the Tool Axis symbol then click the Reverse Direction button in the Tool Axis dialog box.

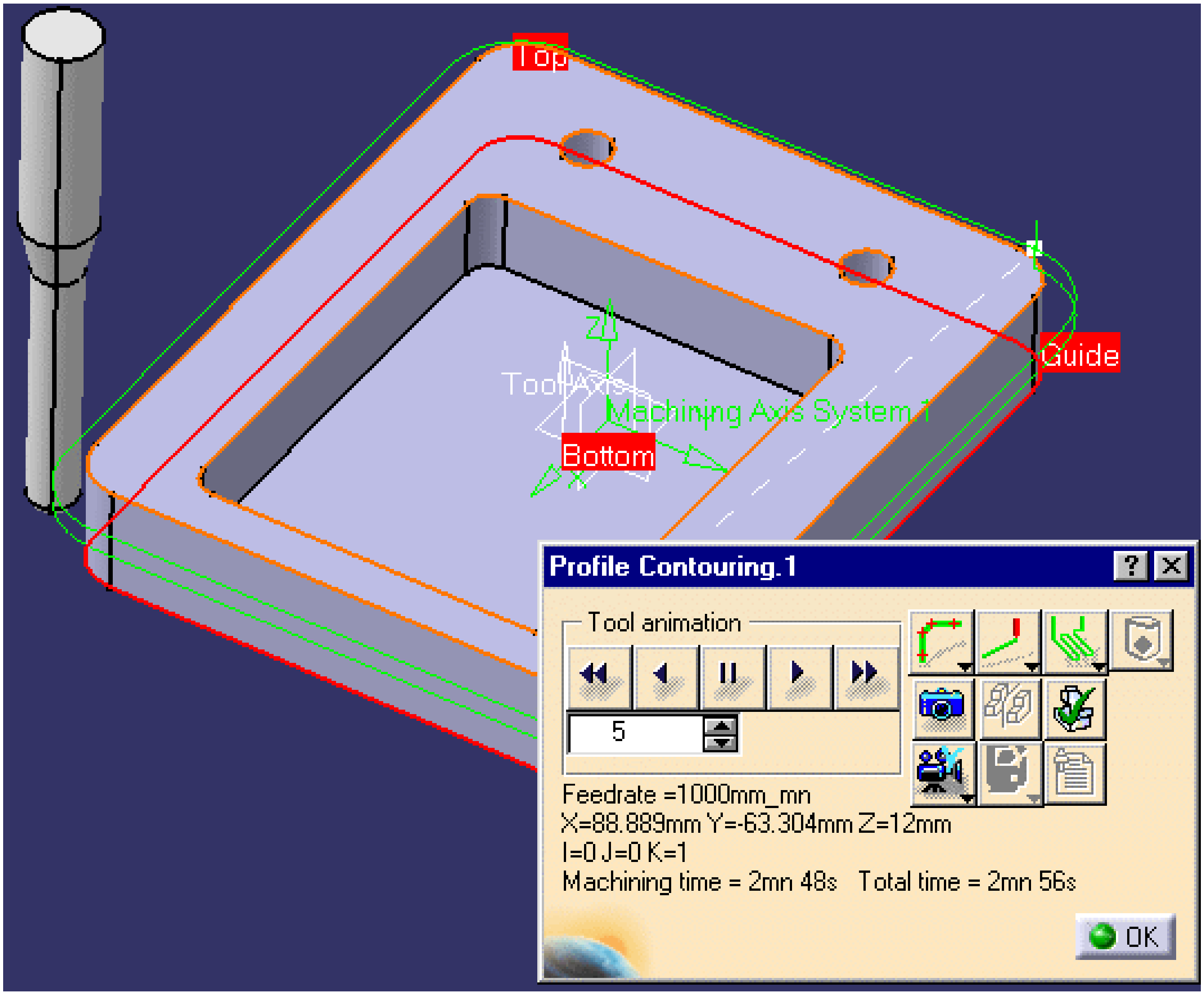
   You can display the tool with the specified orientation by selecting the Display tool checkbox.



8. Click **Preview** in the dialog box to request that the program verifies the parameters that you have specified.

   A message box appears giving feedback about this verification.

9. Click Replay in the dialog box to visually check the operation's tool path.

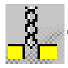


At the end of the replay click OK to return to the Profile Contouring dialog box.

10. Click OK to create the operation.

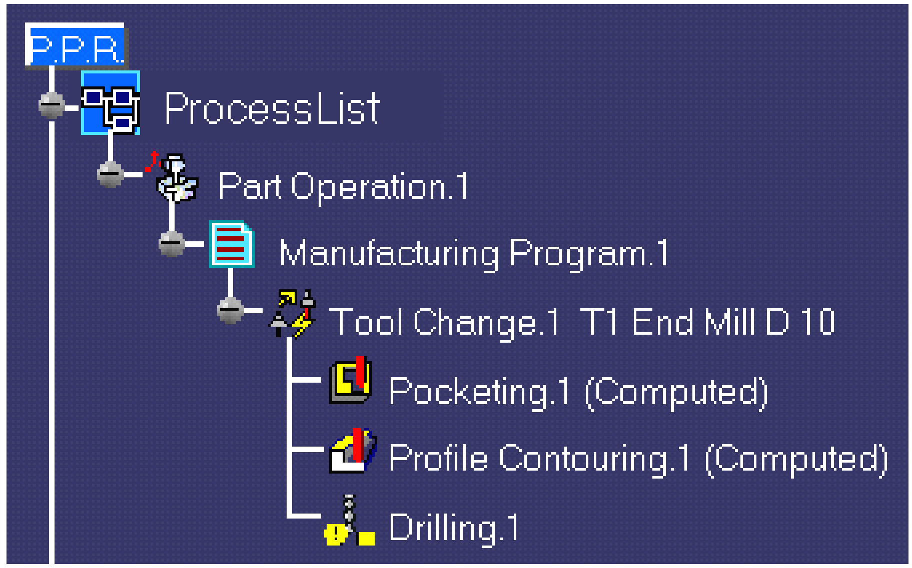
Create a Drilling Operation

This task shows you how to insert a drilling operation in the program.

1. Select the Drilling icon .

   The Drilling dialog box appears directly at the Geometry page .

   The program is updated to include a Drilling operation.



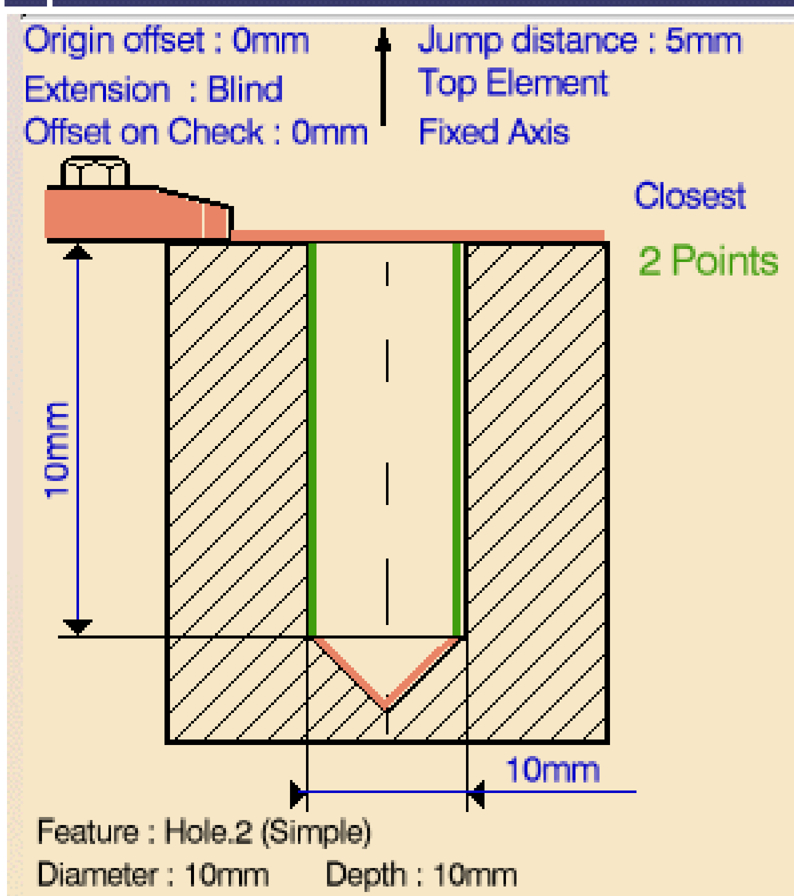
   The Drilling dialog box appears.

2. Select the red hole depth representation in the sensitive icon.

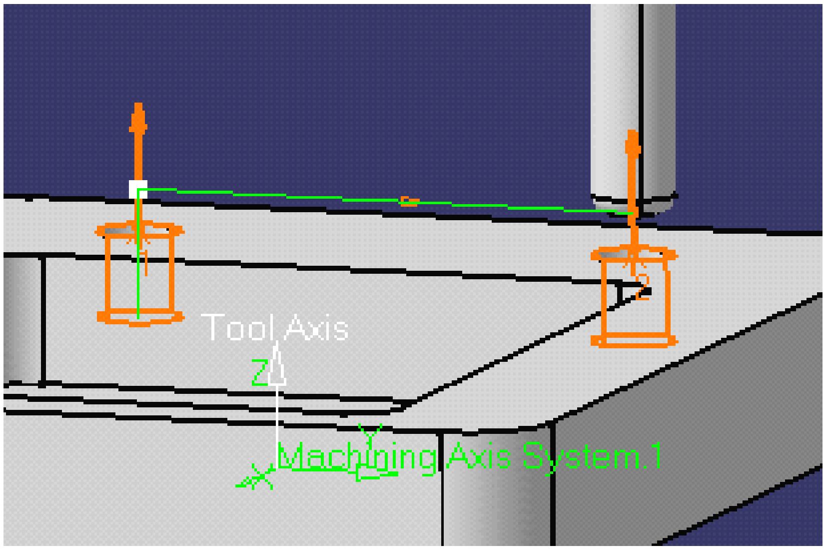
   The Pattern Selection dialog box appears to help you specify the pattern of holes to be machined.
3. Select the cylindrical feature of the first hole.
4. Select the second hole feature, then double click to end hole selection.

   The Drilling dialog box replaces the Pattern Selection dialog box.

   The icon is updated with geometric information about the first selected hole of the pattern.



5. Double click the Jump distance parameter in the sensitive icon, then enter a value of 5mm in the Edit Parameter dialog box.
6. Click Replay to replay the operation as described previously.

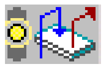


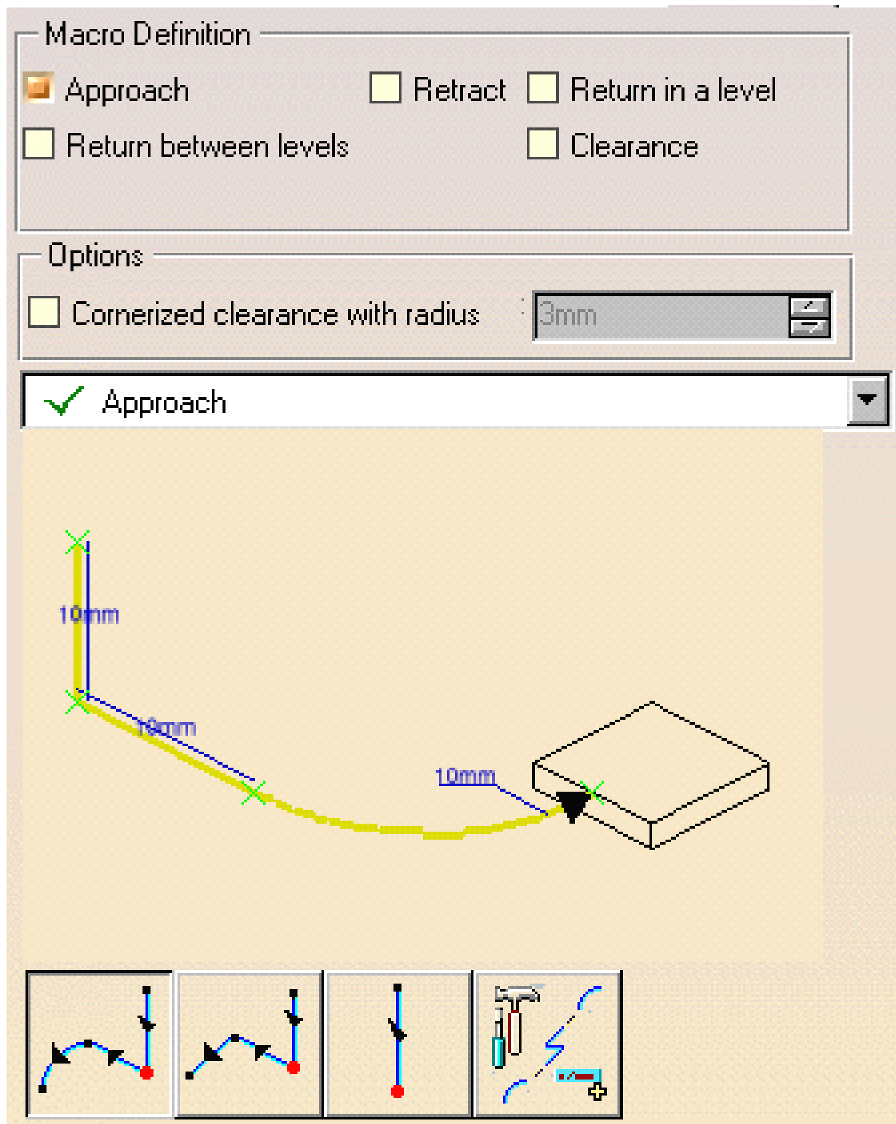
Click OK to return to the Drilling dialog box.

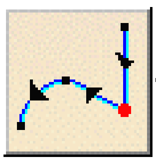
7. Click OK to create the Drilling operation in the program.

Assign a Macro

This task shows you how to assign a circular approach macro to the Profile Contouring operation.

1. Double click the Profile Contouring operation in the program, then select the Macro tab page .



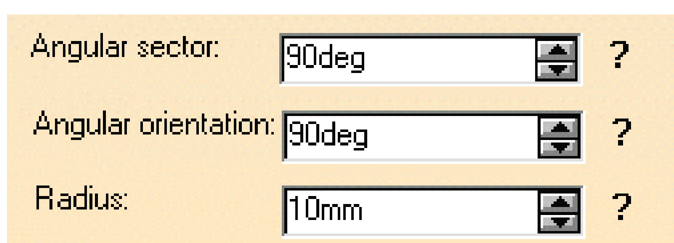
2. Select the Approach checkbox.
3. Click the Circular Approach icon .

   An icon representing the approach motion is displayed.

   Default values are displayed on the individual paths of the macro.

4. Double click the circular path in the icon.

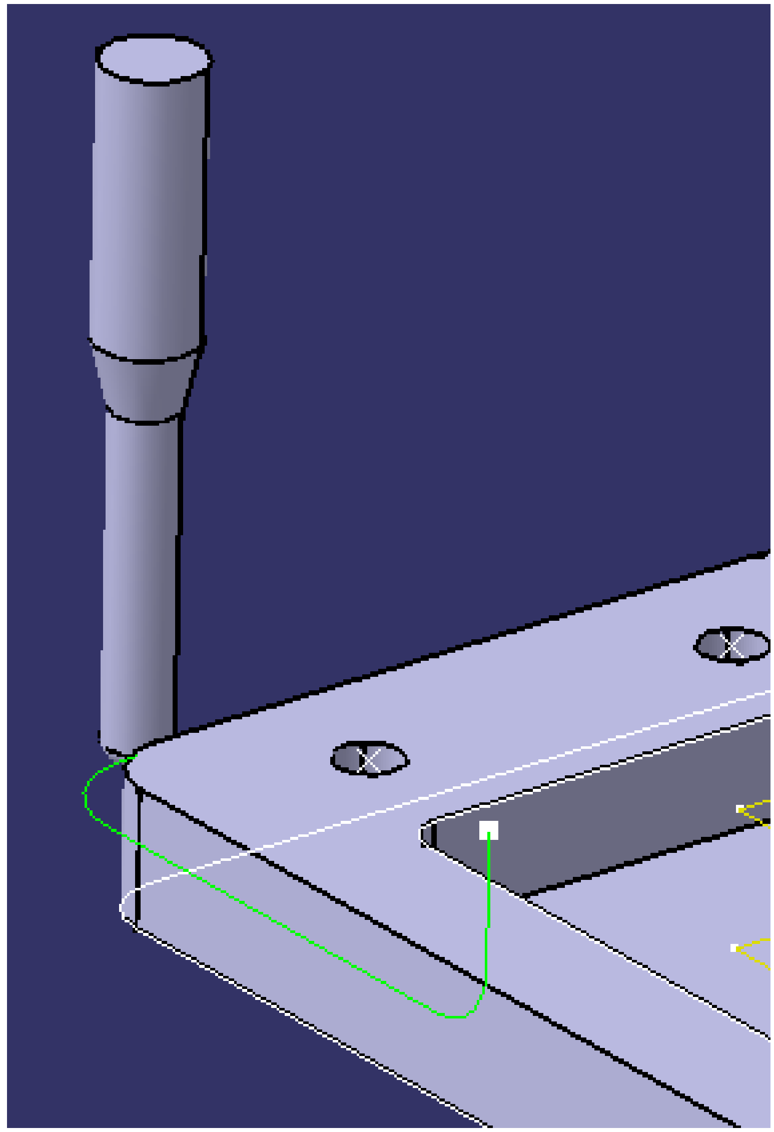
   A dialog box appears allowing you to specify the desired parameters of the circular path.



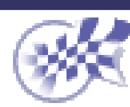
5. Enter values in the dialog box as shown and click OK.

6. Click Replay in the Profile Contouring dialog box to verify the approach motion.

   At the end of the replay click OK to return to the Profile Contouring dialog box.

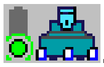


7. Click OK to assign the specified macro to the operation.

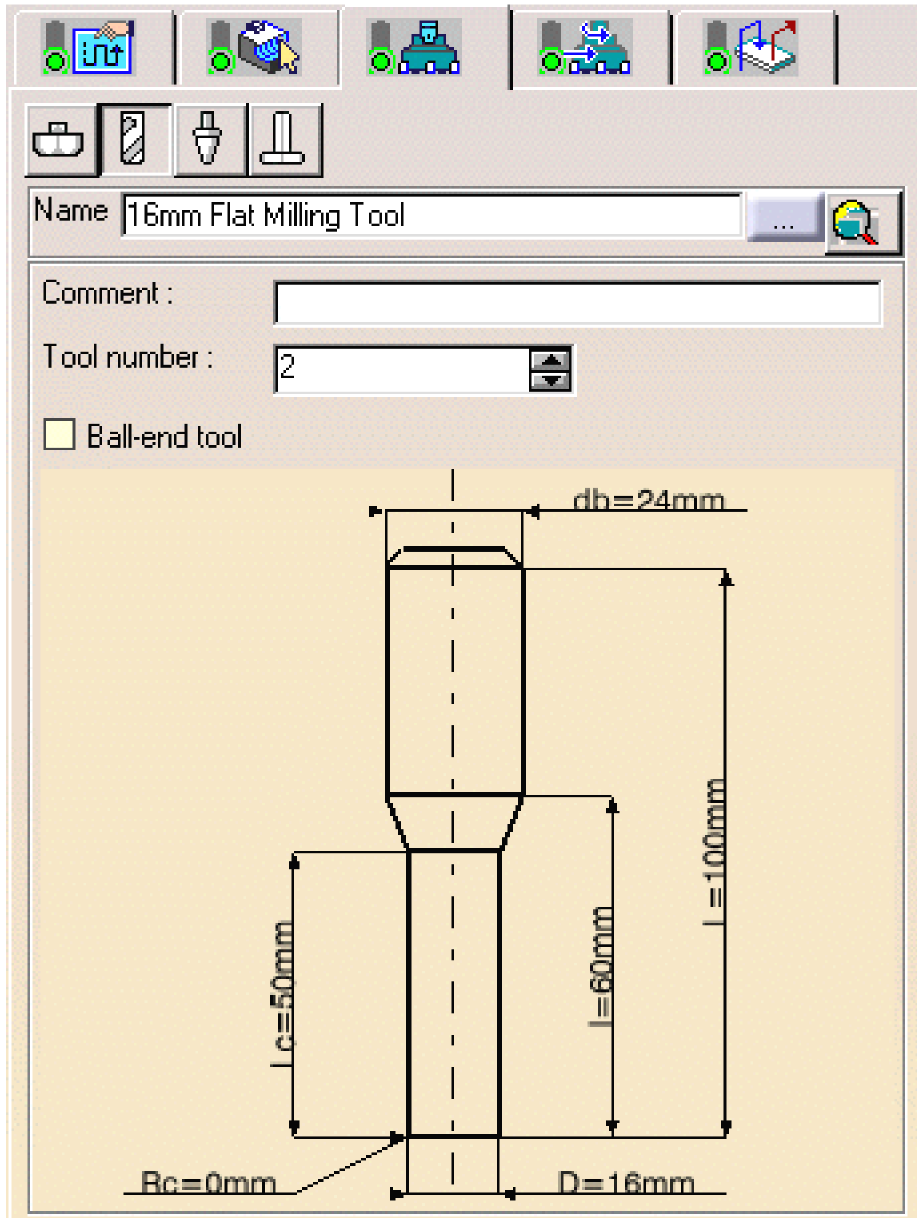


Assign a Tool

This task shows you how to assign another tool to an operation.

1. Double click the Profile Contouring operation in the program, then select the Tool tab page .
2. Enter a name of the new tool (for example, 16mm Flat Milling Tool).
3. Double click the D (nominal diameter) parameter in the icon, then enter 16mm in the Edit Parameter dialog box.

   The tool icon is updated to take the new value into account.



4. Double click the Rc (corner radius) parameter in the icon, then enter 0mm in the Edit Parameter dialog box.

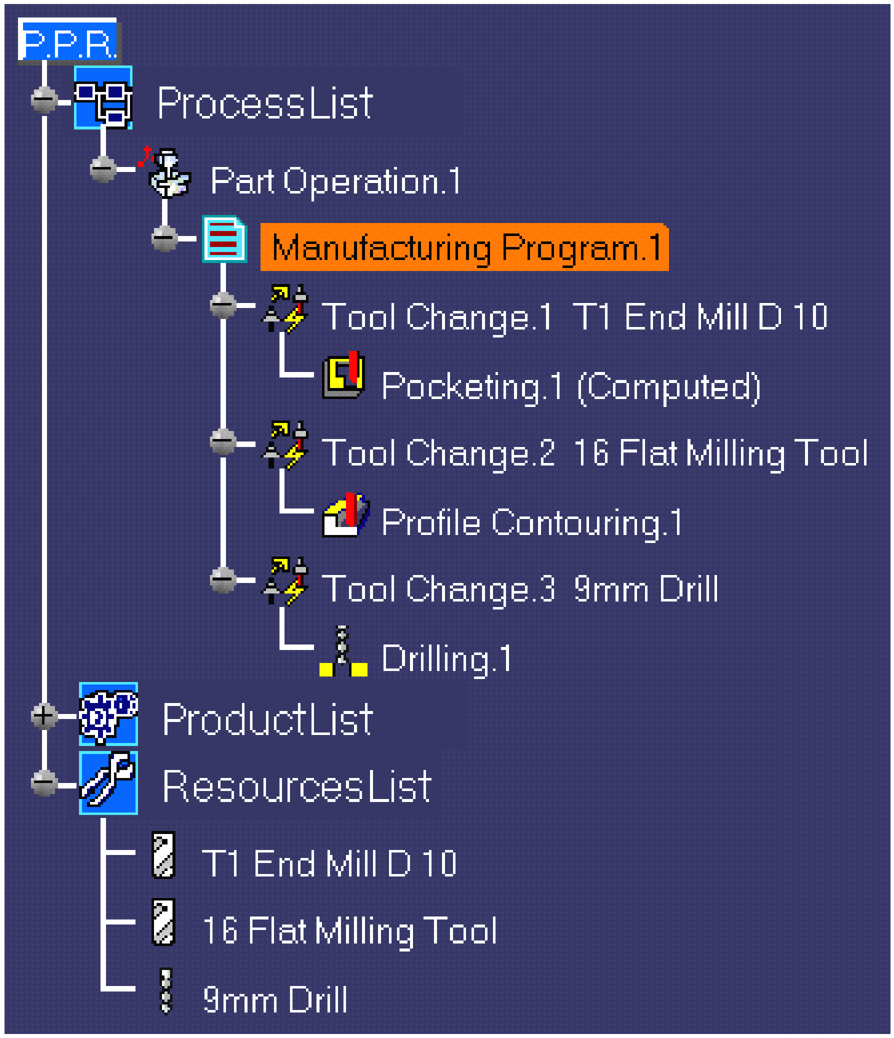
   Set the db (body diameter) parameter to 24mm in the same way.

   The Tool number is set to 2.

5. Click OK to accept the new tool.

The program is automatically updated.

You can modify the tools of the other operations in the same way. For example, you may want to replace the End Mill by a Drill in the Drilling operation.

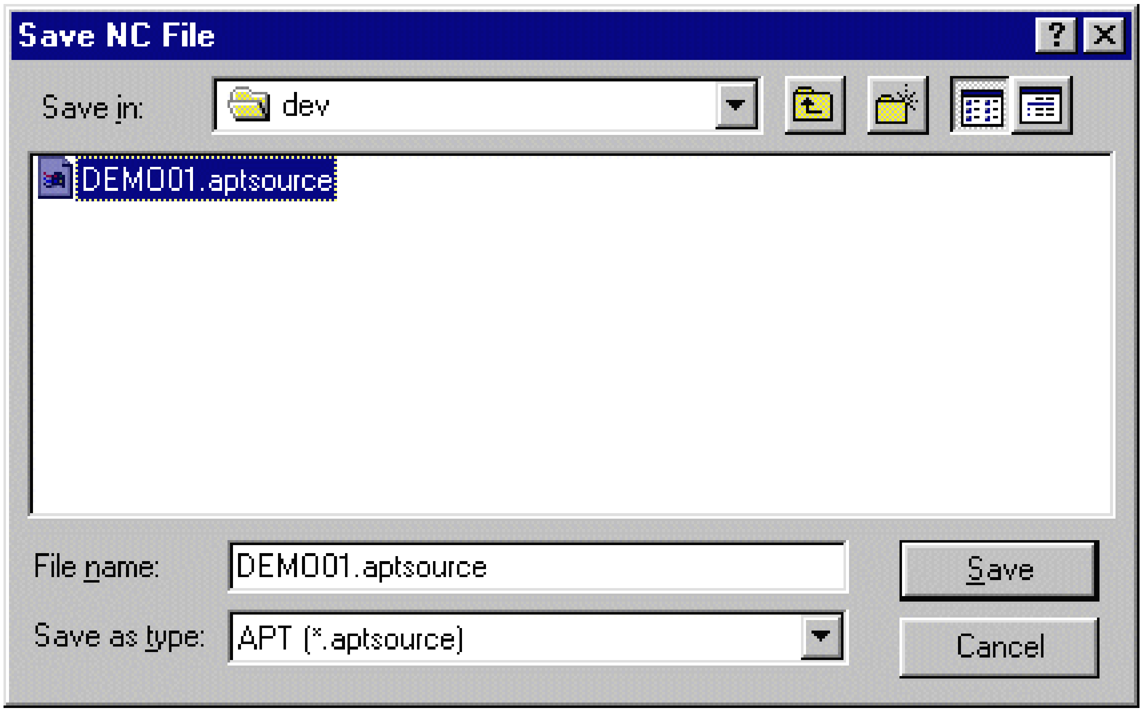

P.P.R.
ProcessList
Part Operation.1
Manufacturing Program.1
Tool Change.1 T1 End Mill D 10
Pocketing.1 (Computed)
Tool Change.2 16 Flat Milling Tool
Profile Contouring.1
Tool Change.3 9mm Drill
Drilling.1
ProductList
ResourcesList
T1 End Mill D 10
16 Flat Milling Tool
9mm Drill

Generate NC Code

This task shows you how to generate the NC code from the program.

1. Use the right mouse key on the Manufacturing Program.1 entity in the tree to select Manufacturing Program.1 object > Generate NC Code Interactively.

   The Save NC File dialog box appears.



2. Select the folder where you want the file to be saved and specify the name of the file.
3. Click Save to create the APT file.

Here is an extract from a typical APT source file that could be generated:

```
$$ -----------------------------------------------------------------
$$ Generated on Thursday, May 10, 2001 04:58:20 PM
$$ -----------------------------------------------------------------
$$ Manufacturing Program.1
$$ Part Operation.1
$$*CATIA0
$$ Manufacturing Program.1
$$ 1.00000 0.00000 0.00000 0.00000
$$ 0.00000 1.00000 0.00000 0.00000
$$ 0.00000 0.00000 1.00000 0.00000
PARTNO PART TO BE MACHINED
COOLNT/ON
CUTCOM/OFF
PPRINT OPERATION NAME : Tool Change.1
$$ Start generation of : Tool Change.1
TLAXIS/ 0.000000, 0.000000, 1.000000
$$ TOOLCHANGEBEGINNING
RAPID
GOTO / 0.00000, 0.00000, 100.00000
```

```
CUTTER/ 10.000000, 2.000000, 3.000000, 2.000000, 0.000000$
, 0.000000, 100.000000
TOOLNO/1, 10.000000
TPRINT/T1 End Mill D 10
LOADTL/1
$$ End of generation of : Tool Change.1
PPRINT OPERATION NAME : Pocketing.1
$$ Start generation of : Pocketing.1
FEDRAT/ 1000.0000,MMPM
SPINDL/ 70.0000,RPM,CLW
GOTO / 30.61644, 2.50000, 5.00000
GOTO / 17.50000, 2.50000, 5.00000
...
GOTO / 30.61644, 35.00000, 5.00000
$$ End of generation of : Pocketing.1
PPRINT OPERATION NAME : Tool Change.2
$$ Start generation of : Tool Change.2
$$ TOOLCHANGEBEGINNING
RAPID
GOTO / 0.00000, 0.00000, 100.00000
CUTTER/ 16.000000, 0.000000, 8.000000, 0.000000, 0.000000$
, 0.000000, 100.000000
TOOLNO/2, 16.000000
TPRINT/16 Flat Milling Tool
LOADTL/2
$$ End of generation of : Tool Change.2
PPRINT OPERATION NAME : Profile Contouring.1
$$ Start generation of : Profile Contouring.1
FEDRAT/ 300.0000,MMPM
SPINDL/ 70.0000,RPM,CLW
GOTO / -69.00000, 40.00000, 46.00000
...
GOTO / -69.00000, 50.00000, 0.00000
$$ End of generation of : Profile Contouring.1
PPRINT OPERATION NAME : Tool Change.3
$$ Start generation of : Tool Change.3
$$ TOOLCHANGEBEGINNING
RAPID
GOTO / 0.00000, 0.00000, 100.00000
CUTTER/ 9.000000, 0.000000, 4.500000, 2.598076, 30.000000$
, 0.000000, 100.000000
TOOLNO/3, 9.000000
TPRINT/9mm Drill
LOADTL/3
$$ End of generation of : Tool Change.3
PPRINT OPERATION NAME : Drilling.1
$$ Start generation of : Drilling.1
LOADTL/3,1
SPINDL/ 70.0000,RPM,CLW
RAPID
GOTO / -40.00000, -30.00000, 25.00000
RAPID
GOTO / -40.00000, -30.00000, 21.00000
CYCLE/DRILL, 10.000000, 1.000000, 1000.000000,MMPM
GOTO / -40.00000, -30.00000, 20.00000
GOTO / -40.00000, 30.00000, 20.00000
```

```
CYCLE/OFF
RAPID
GOTO / -40.00000, 30.00000, 21.00000
RAPID
GOTO / -40.00000, 30.00000, 25.00000
$$ End of generation of : Drilling.1
SPINDL/OFF
REWIND/0
END
```

Basic Tasks

The basic tasks you will perform in the Prismatic Machining workbench involve creating, editing and managing machining operations and other entities of the NC manufacturing process.

Milling Operations

The tasks in this section show you how to create 2.5 axis milling operations in your NC manufacturing program.

Pocketing Operations

Select the Pocketing icon then select the geometry to be machined (open or closed pocket, islands, and so on). Specify the tool to be used.
Set parameters for axial and radial machining and other criteria such as finishing and high-speed milling. Set feeds and speeds and NC macros as needed.

A Pocketing operation can be created for machining:

- Closed pockets

  Tool machines the area delimited by hard boundaries
- Open pockets

  Tool machines the area that has a least one soft boundary.

Note that **P2 functionalities** for Pocketing include Automatic Draft Angle, all Finishing parameters, and Sectioning for guiding element selection.
To edit in **P1** a Pocketing operation that was created in **P2,** the following parameter values must be set:

- Automatic draft angle = 0 deg
- Finishing Mode = No finish path
- Side finish thickness = 0.0 mm
- Side finish thickness on bottom = 0.0 mm
- Bottom finish thickness = 0.0 mm
- Spring pass = no
- Avoid scallops on bottom = no
- HSM Cornering on side finish path = no
- HSM Corner radius = 1 mm
- HSM Limit angle = 10 deg.

Facing Operations

Create a Facing Operation: Select the Facing icon then select the geometry to be machined and specify the tool to be used.
Set parameters for axial and radial machining and other criteria such as finishing and high-speed milling. Set feeds and speeds and NC macros as needed.

Note that **P2 functionalities** for Facing include all Finishing parameters and Sectioning for guiding element selection.
To edit in **P1** a Facing operation that was created in **P2,** the following parameter values must be set:

- Finishing Mode = No finish path
- Bottom finish thickness = 0.0 mm
- HSM Cornering on side finish path = no
- HSM Corner radius = 1 mm
- HSM Limit angle = 10 deg.

Profile Contouring Operations

Select the Profile Contouring icon then select the geometry to be machined and specify the tool to be used.
Set parameters for axial and radial machining and other criteria such as finishing.
Set feeds and speeds and NC macros as needed.

A Profile Contouring operation can be created for machining:

- Between two planes
  Tool follows contour between top and bottom planes while respecting user-defined geometry limitations and machining strategy parameters.
- Between two curves **(P2 functionality)**
  Tool follows trajectory defined by top and bottom guide curves while respecting user-defined geometry limitations and machining strategy parameters.
- Between a curve and surfaces **(P2 functionality)**
  Tool follows trajectory defined by a top guide curve and bottom surfaces while respecting user-defined geometry limitations and machining strategy parameters.
- By flank contouring **(P2 functionality)**
  Tool flank machines vertical part surface while respecting user-defined geometry limitations and machining strategy parameters.

Note that **P2 functionalities** for Profile Contouring include Automatic Draft Angle, all Finishing parameters, and Sectioning for guiding element selection.
To edit in **P1** a Profile Contouring operation that was created in **P2,** the following parameter values must be set:

- Automatic draft angle = 0 deg
- Finishing Mode = No finish path
- Side finish thickness = 0.0 mm
- Side finish thickness on bottom = 0.0 mm
- Bottom finish thickness = 0.0 mm
- Spring pass = no.

Point to Point Operations

Create a Point to Point Operation: Select the Point to Point icon then define a sequence of elementary GOTO and PS/DS/CS tool motions. Specify the tool to be used, machining parameters, NC macros, and feeds and speeds as needed.

Curve Following Operations

Create a Curve Following Operation: Select the Curve Following icon then select the geometry to be machined and specify the tool to be used. Specify machining parameters and feeds and speeds as needed.

Operations for Reworking Corners and Channels

Corners and channels left unmachined by Pocketing or Profile Contouring operations can be identified thanks to a Prismatic Rework Area feature. This feature can then be used to Create operations for reworking corners and channels.

Create a Pocketing Operation for Machining Closed Pockets

This task shows how to insert a Pocketing operation in the program when the pocket to be machined comprises hard boundaries only (that is, a closed pocket).

To create the operation you must define:

- the Pocketing mode as Closed Pocket
- the geometry of the pocket to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

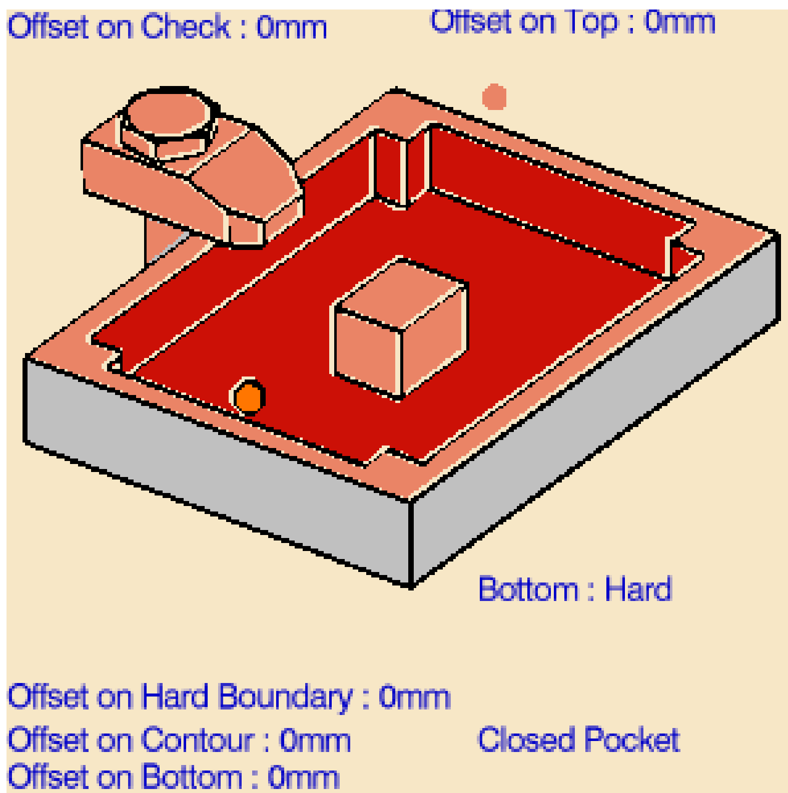
Open the PrismaticMilling01.CATPart document, then select NC Manufacturing > Prismatic Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Pocketing icon.

   A Pocketing entity along with a default tool is added to the program.

   The Pocketing dialog box appears directly at the Geometry tab page.

   This tab page includes a sensitive icon to help you specify the geometry to be machined.



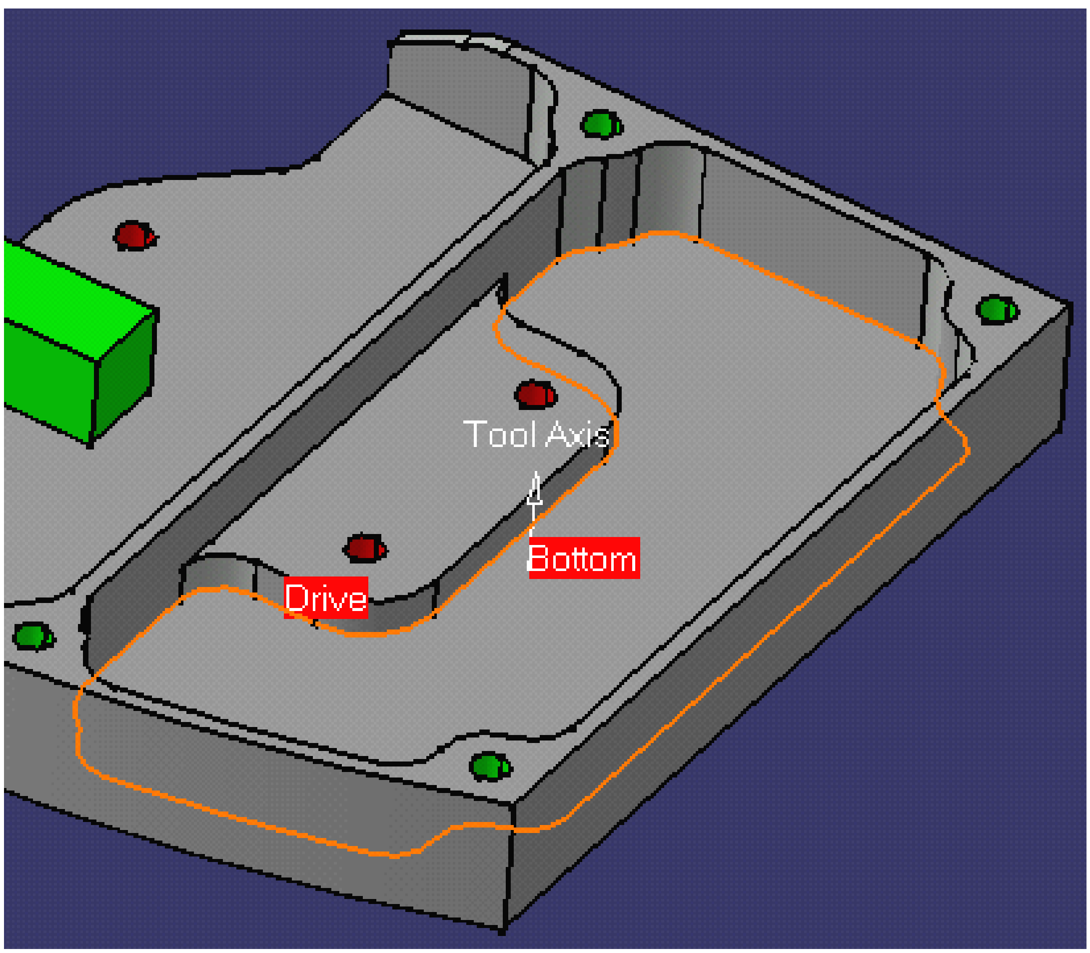
The bottom and flanks of the icon are colored red indicating that this geometry is required for defining the pocket.
All other pocket geometry is optional.

Make sure that the Pocketing style is set to **Closed Pocket**.

2. Right click the red Bottom in the icon and select **Contour Detection** from the contextual menu.

   Click the red Bottom then select the desired pocket bottom in the 3D window.

   The pocket boundary is automatically deduced thanks to the Contour Detection setting. This is indicated by the highlighted Drive elements.



The bottom and flanks of the icon are now colored green indicating that this geometry is now defined.

For parts containing islands, you can right click the red Bottom in the icon and select **Island Detection** from the contextual menu. This allows island boundaries to be deduced automatically.

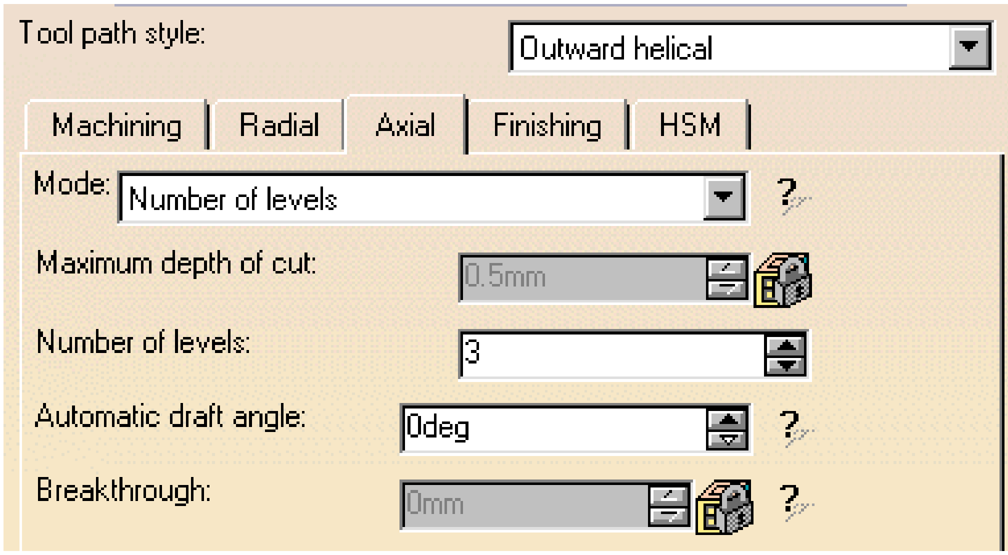
3. Click the Top Plane in the icon then select the desired top element in the 3D window.
4. Set the following offsets:
   - 1.5mm on hard boundary
   - 0.25mm on bottom.

If your part includes islands you can specify different offsets on individual islands using the contextual menu.

If you specify an Offset on Contour, it is added to any defined Offset on Hard Boundary and Offset on Islands.

You can select a **Start point** and an **End point** as preferential start and end positions for the operation. This allows better control for optimizing the program according to the previous and following operations.

5. Select the Strategy tab page and choose the desired tool path style: Inward helical, Outward helical or Back and forth.



You can then use the tab pages to set parameters for:

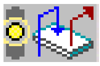
- machining such as machining tolerance
- radial strategy
- axial strategy (number of levels = 3, for example)
- pocket finishing
- high speed milling.

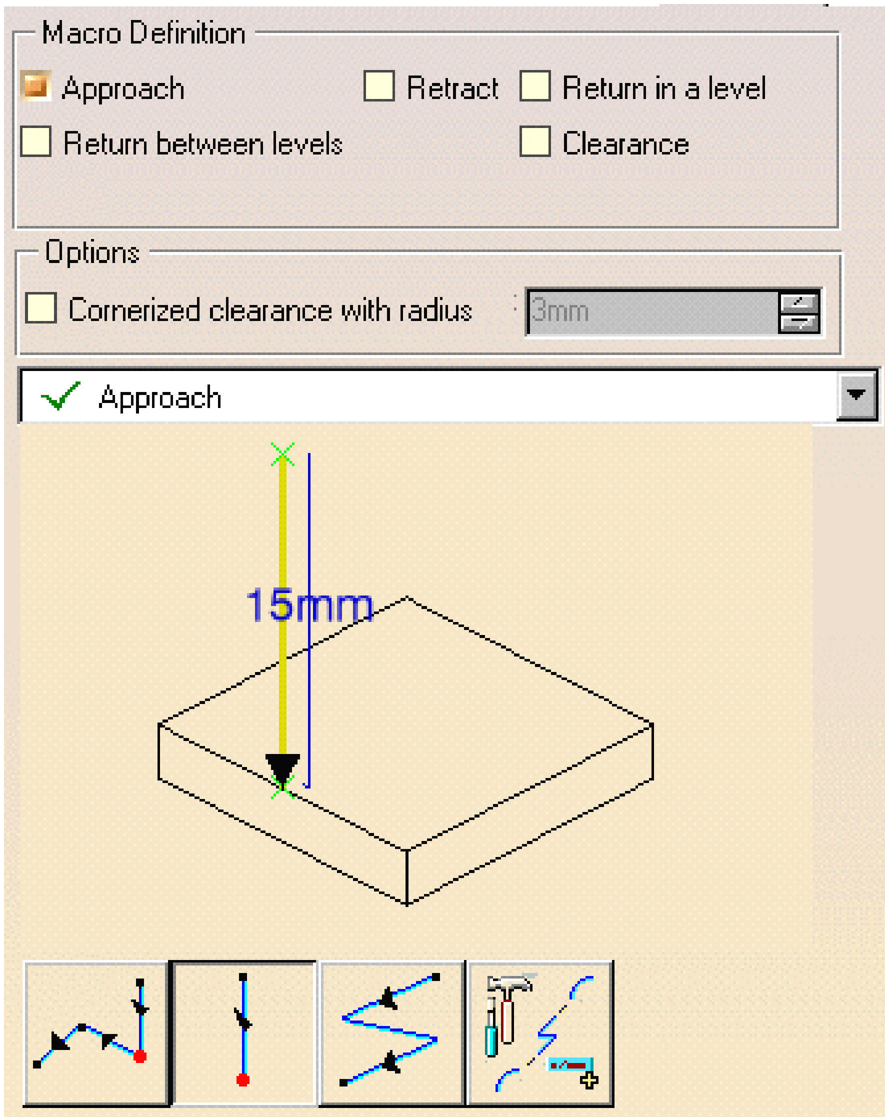
For machining a multi-domain pocket using a helical tool path style, you can select **Always stay on bottom** to avoid unnecessary linking transitions. This option forces the tool to remain in contact with the pocket bottom when moving from one domain to another.
In this case, you can also select **Inward/outward mix** to authorize changing from one helical style from one pocket domain to another.

A tool is proposed by default when you create a machining operation. If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

6. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

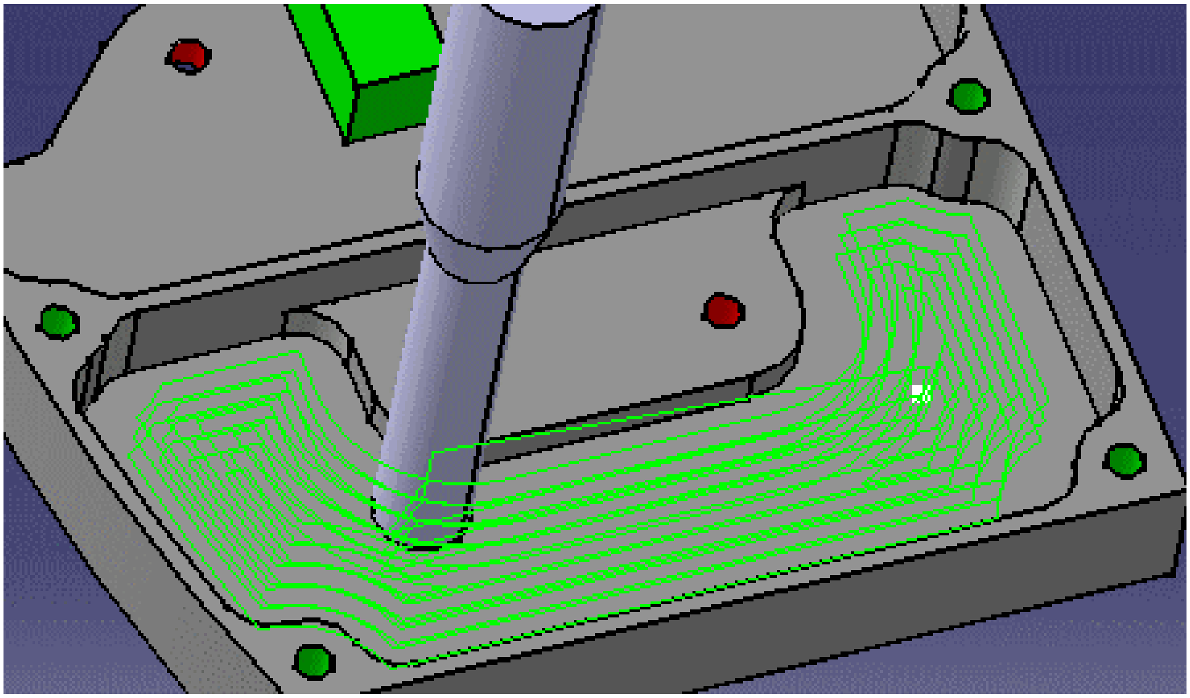
7. Select the Macros tab page  to specify the operation's transition paths (approach and retract motion, for example).



- Select the Approach checkbox
- Select an Approach macro icon to specify the desired type of approach motion (linear, for example). A sensitive icon appears with a representation of the macro.
- Double click the distance parameter in the sensitive icon and enter the desired value in the pop-up dialog box.
- Repeat this procedure to specify the Retract motion.

See Define Macros of an Operation for another example of specifying transition paths on a machining operation.

Before accepting the operation, you should check its validity by replaying the tool path.

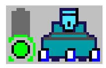
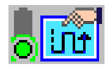


8. Click OK to create the operation.

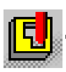
Create a Pocketing Operation for Machining Open Pockets

This task shows how to insert a Pocketing operation in the program when the pocket to be machined comprises at least one soft boundary (that is, an open pocket).

To create the operation you must define:

- the Pocketing mode as Open Pocket
- the geometry of the pocket to be machined 
- the tool that will be used 
- the parameters of the machining strategy 
- the feedrates and spindle speeds 
- the macros (transition paths) .

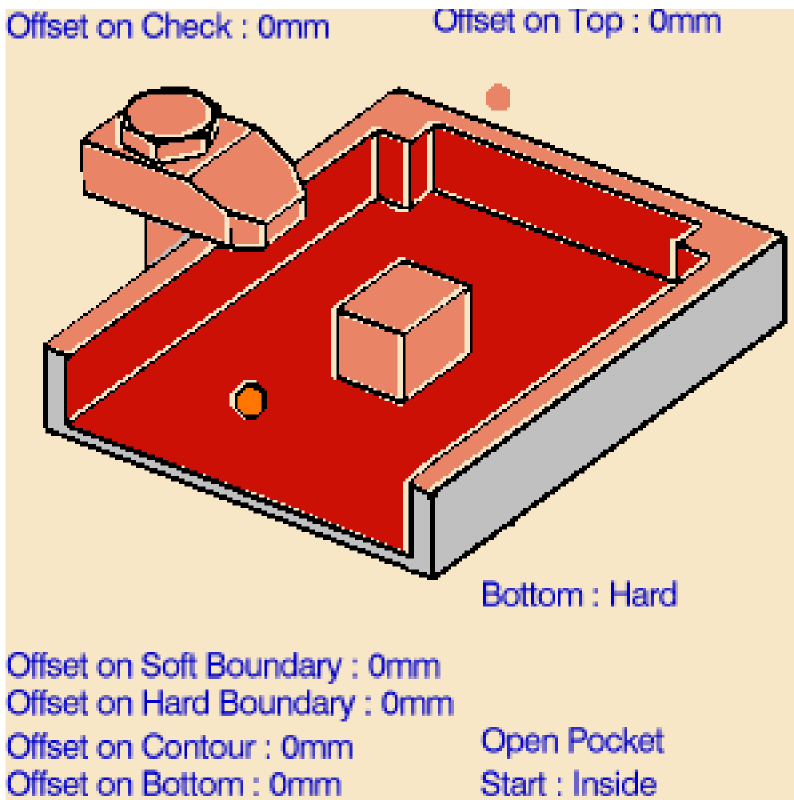
Open the PrismaticMilling02.CATPart document, then select NC Manufacturing > Prismatic Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Pocketing icon .

   A Pocketing entity along with a default tool is added to the program.

   The Pocketing dialog box appears directly at the Geometry tab page .

   This tab page includes a sensitive icon to help you specify the geometry to be machined.



The bottom and flanks of the icon are colored red indicating that this geometry is required for defining the pocket. All other pocket geometry is optional.

Make sure that the Pocketing style is set to **Open Pocket**.

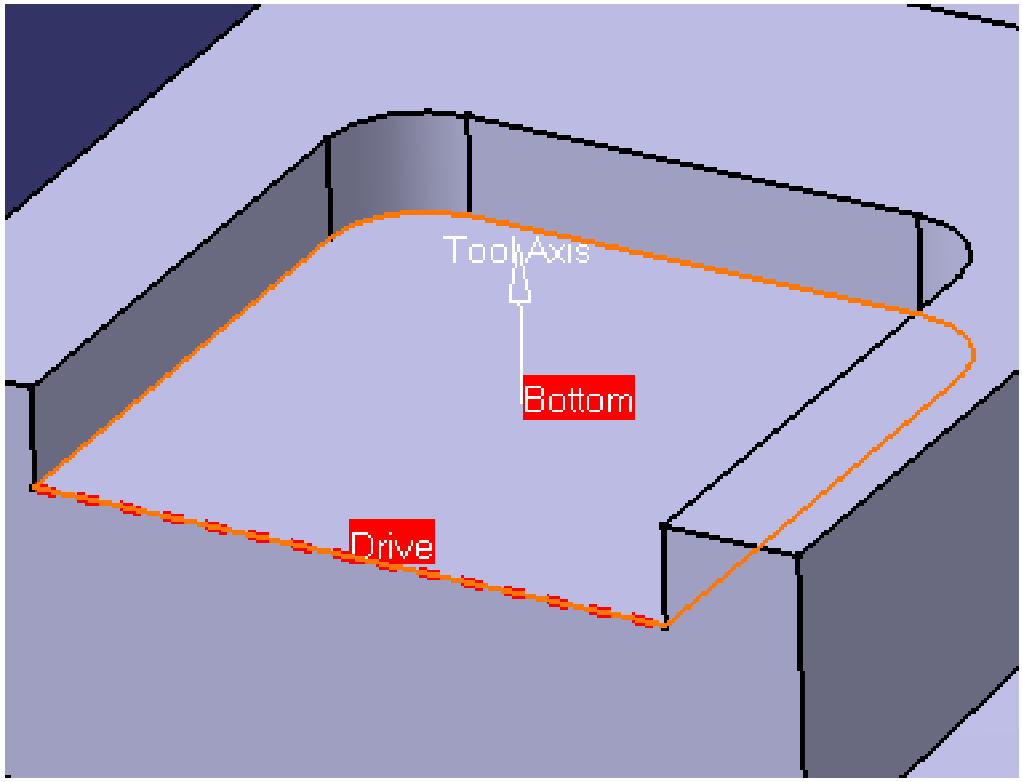
2. Right click the red Bottom in the icon and select **Contour Detection** from the contextual menu.

   Click the red Bottom then select the desired pocket bottom in the 3D window.

   The pocket boundary is automatically deduced thanks to the Contour Detection setting. This is indicated by the highlighted Drive elements.

   Hard boundaries are shown by full lines and soft boundaries by dashed lines.

   For edge selection only, you can change a boundary segment from hard to soft (or from soft to hard) by selecting the corresponding edge.



The bottom and flanks of the icon are now colored green indicating that this geometry is now defined.

For parts containing islands, you can right click the red Bottom in the icon and select **Island Detection** from the contextual menu. This allows island boundaries to be deduced automatically.

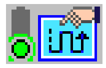
3. Click the Top Plane in the icon then select the desired top element in the 3D window.
4. Set the following offsets:
   - 1.5mm on hard boundary
   - 0.25mm on bottom.

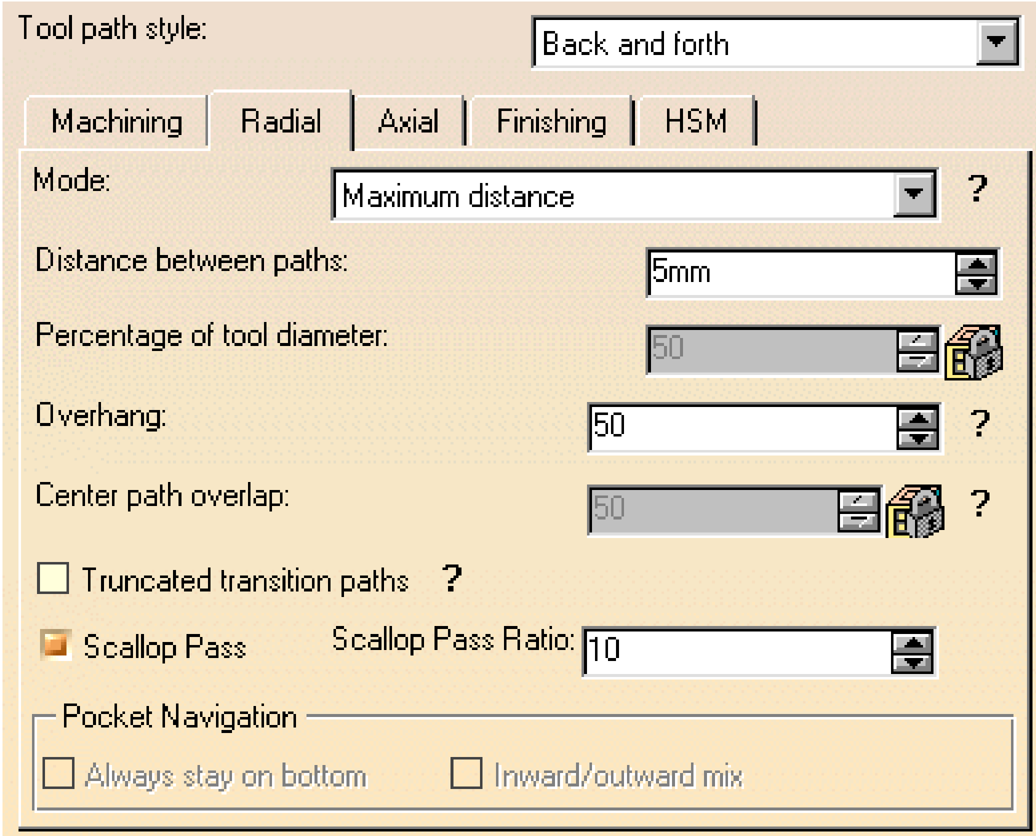
If your part included islands you could specify different offsets on individual islands using the contextual menu.

If you specify an Offset on Contour, it is added to any defined Offset on Hard Boundary and Offset on Islands.

You can select a **Start point** and an **End point** as preferential start and end positions for the operation. This allows better control for optimizing the program according to the previous and following operations.

Note that the Start point can be located outside the pocket. In this case, you must specify a clearance with respect to the pocket boundary.

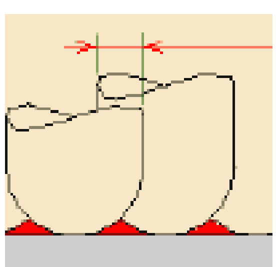
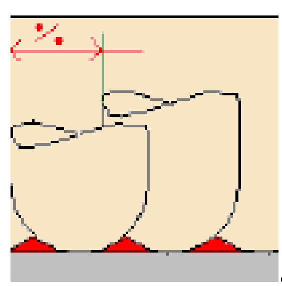
5. Select the Strategy tab page  and choose the desired tool path style: Inward helical, Outward helical or Back and forth.



You can then use the tab pages to set:

- machining parameters such as machining tolerance
- radial mode and associated parameters (overhang = 50, for example)
- axial mode and associated parameters (number of levels = 3, for example)
- pocket finishing parameters
- high speed milling parameters.

The Radial mode determines the distance between two consecutive paths. You should either set a maximum distance value or give a percentage of the tool diameter to be used as:

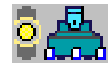
Tool diameter ratio  or Stepover ratio 

.

For pocketing using a Back and Forth tool path style:

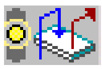
- selecting the **Truncated transition paths** checkbox allows the tool to follow the external profile more exactly by allowing the transition portion of the trajectory to be truncated

- selecting the **Scallop pass** checkbox allows a final machining pass around the exterior of the trajectory for removing scallops. The position of the final pass can be adjusted by means of the **Scallop pass ratio**.

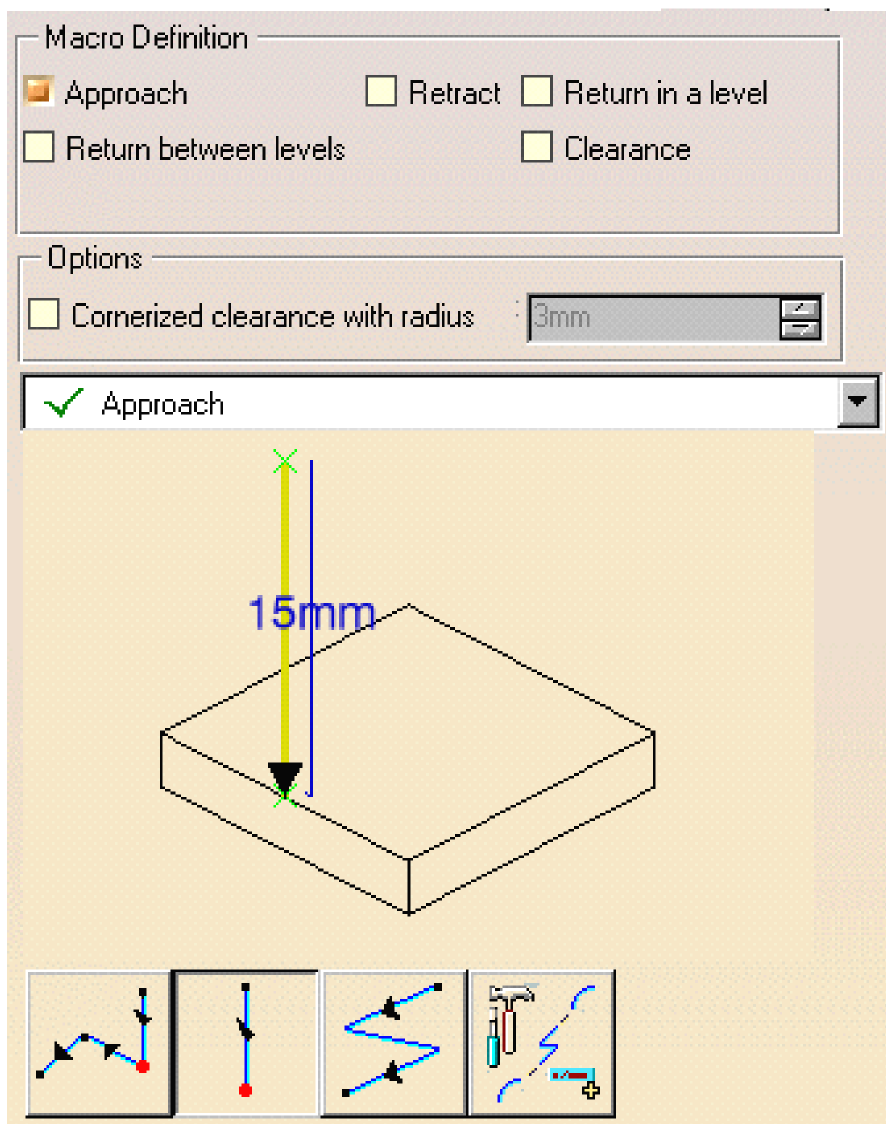
6. A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

   This is described in Edit the Tool of an Operation.

7. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

8. Select the Macros tab page  to specify the operation's transition paths (approach and retract motion, for example).

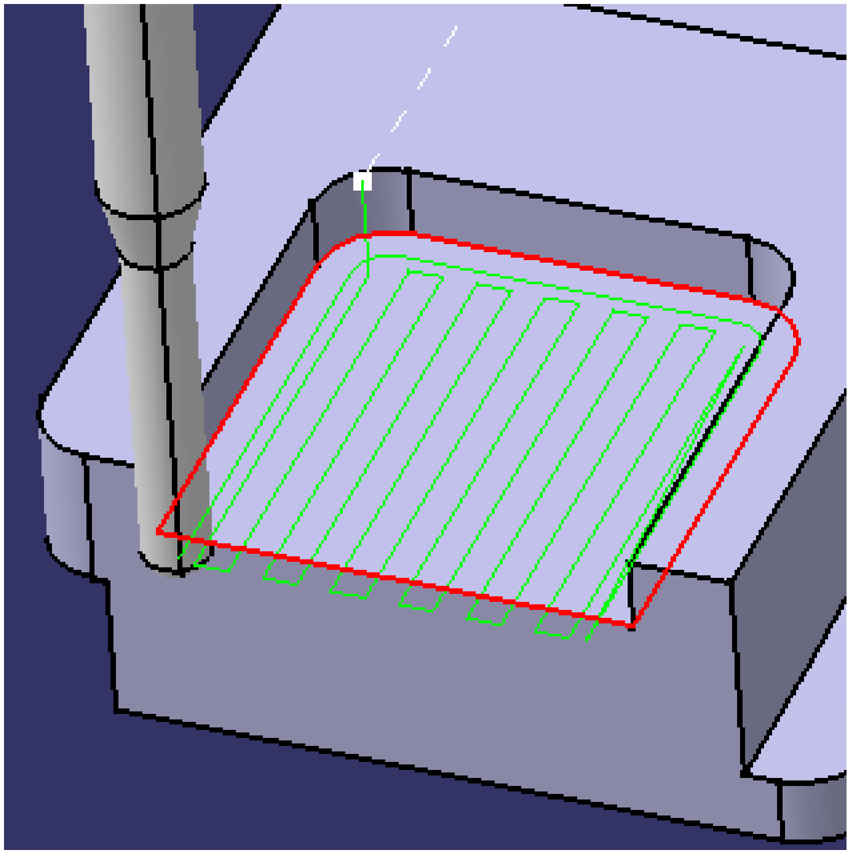
   - Select the Approach checkbox
   - Select an Approach macro icon to specify the desired type of approach motion (linear, for example). A sensitive icon appears with a representation of the macro.
   - Double click the distance parameter in the sensitive icon and enter the desired value in the pop-up dialog box.
   - Repeat this procedure to specify the Retract motion.



   See Define Macros of

an Operation for another example of specifying transition paths on a machining operation.

9. Before accepting the operation, you should check its validity by replaying the tool path.



10. Click OK to create the operation.

Create a Facing Operation

This task shows how to insert a Facing operation in the program.

To create the operation you must define:

- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy :

  the proposed tool path styles are: Inward helical, Back and forth, and One way
- the feedrates and spindle speeds
- the macros (transition paths) .

Open the PrismaticMilling01.CATPart document, then select NC Manufacturing > Prismatic Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

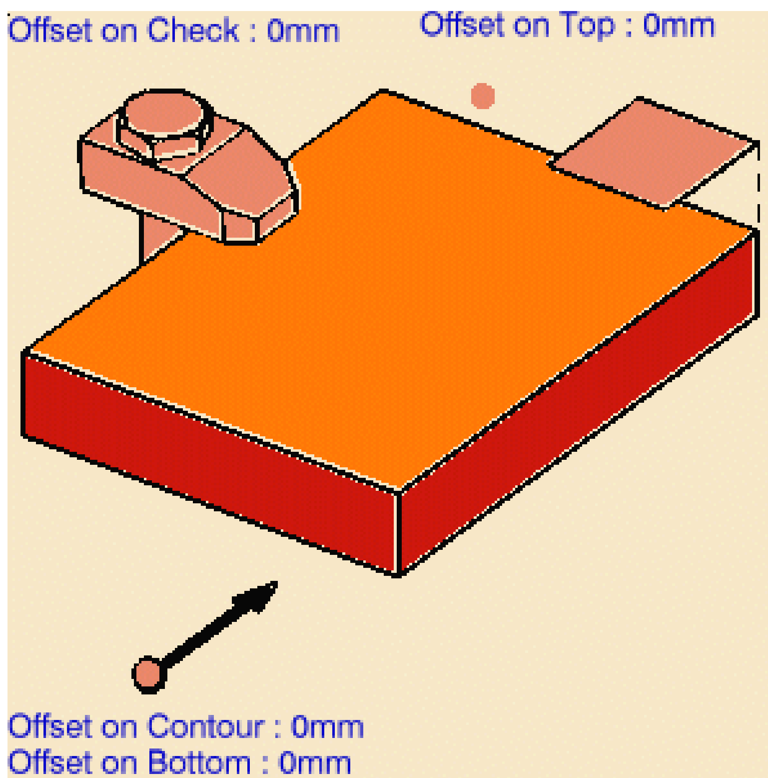
1. Select the Facing icon .

A Facing entity along with a default tool is added to the program.

The Facing dialog box appears directly at the Geometry tab page .

This tab page includes a sensitive icon to help you specify the geometry to be machined.

The part bottom and flanks in the icon are colored red indicating that this geometry is required for defining the operation.
All other geometry is optional.



2. Right click the red Bottom in the icon and select **Contour Detection** from the contextual menu.

   Click the red Bottom then select the underside of the part in the 3D window.

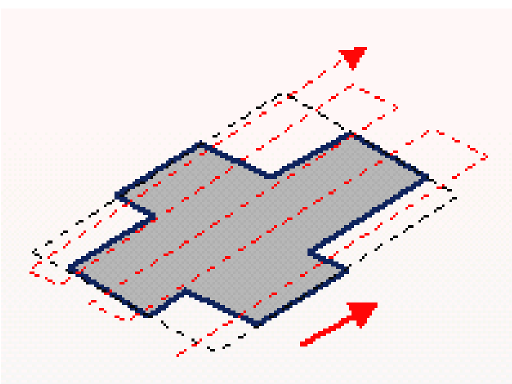
   The part boundary is automatically deduced thanks to the Contour Detection option. This is indicated by the highlighted Drive elements.

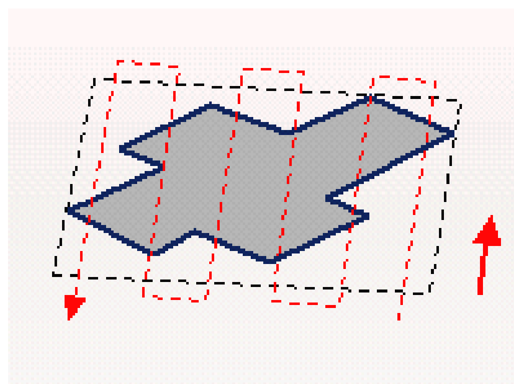
The bottom and flanks of the icon are now colored green indicating that this geometry is now defined.

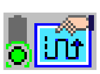
You can select start and end points as preferential start and end positions on the operation. This allows better control for optimizing the program according to the previous and following operations.

For One way and Back and forth tool path styles, you can select the **Bounding envelope** checkbox to machine the maximum bounding rectangle of the part. After selecting the geometry to be machined, this rectangle is computed after defining a machining direction.

The figures below illustrate how machining is done for different machining directions.



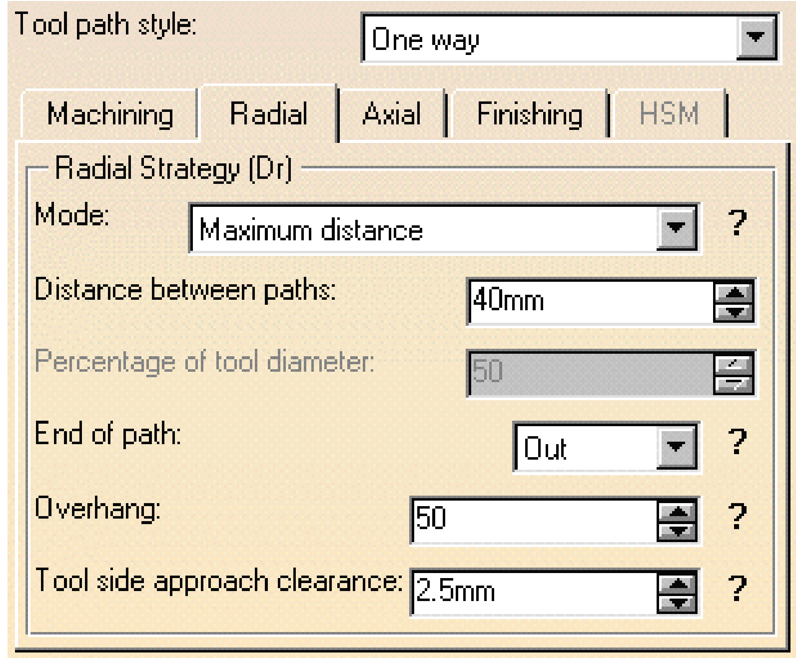


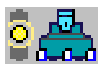
3. Select the Strategy tab page  and choose the desired tool path style: Inward helical, Back and forth, and One way.

   You can then use the tab pages to set parameters for:

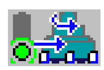
   - machining such as machining tolerance
   - radial strategy (see example)

- axial strategy (number of levels = 1, for example)
- finishing
- high speed milling (for Inward helical tool path style only).

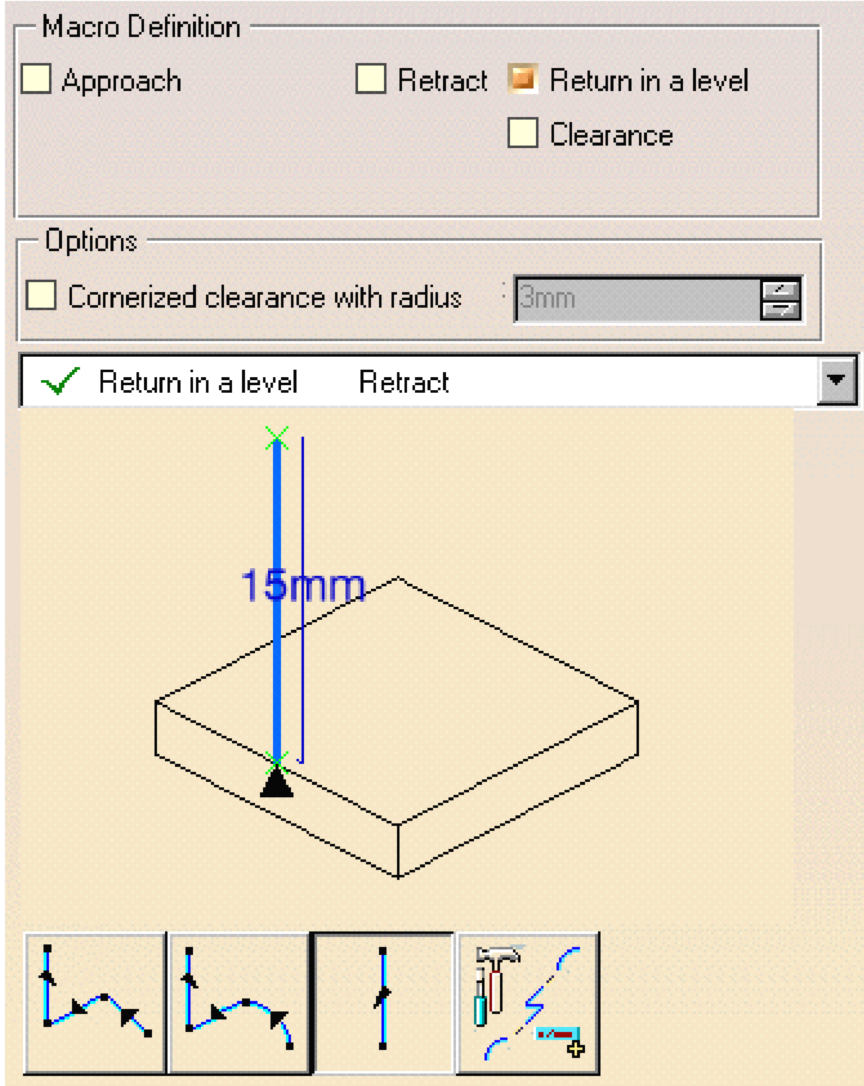


4. Select the Tool tab page  to replace the default tool by a more suitable one.
5. Select the Face Mill icon.

   A 50mm diameter face mill is proposed. You can adjust the parameters as required.

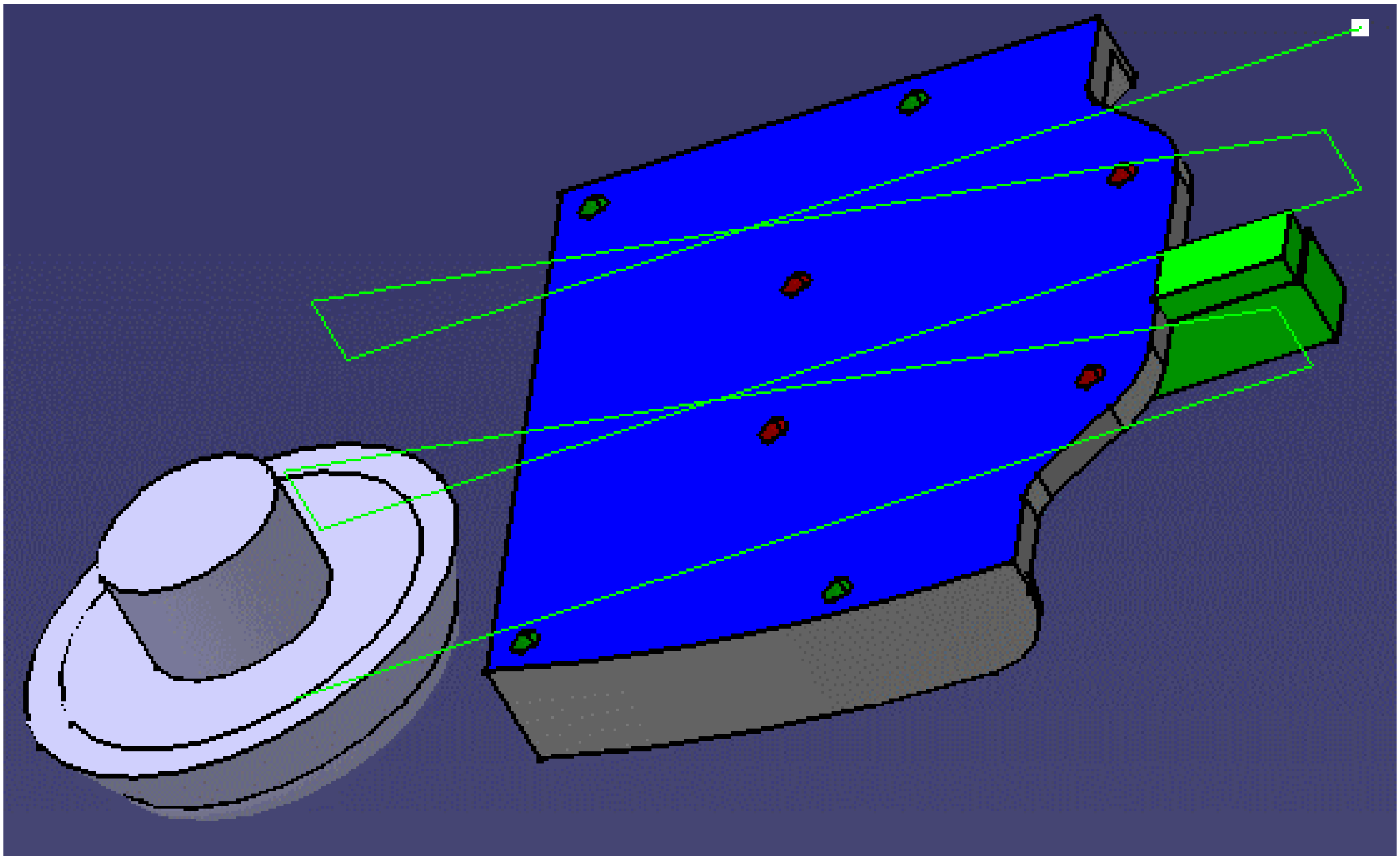
   See Edit the Tool of an Operation for more information about selecting tools.
6. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.
7. Select the Macros tab page  to specify a return macro, which is necessary for the One Way mode.
   - Select the Return in a level checkbox.
   - Select one of the macro icons to specify the desired type of retract motion (linear, for example). A sensitive icon appears with a representation of the path.
   - Double click the distance parameter in the sensitive icon and enter the desired value in the pop-up dialog box.

- Select the Return in a level - Approach in the combo, the repeat the procedure to specify the approach motion.



See Define Macros of an Operation for another example of specifying transition paths on a machining operation.

Before accepting the operation, you should check its validity by replaying the tool path.



8. Click OK to create the operation.

In this scenario the operation used the default start point (that is, the origin of the absolute axis system).

If you want to define a different start point, you can click the start point symbol in the sensitive icon then select a point.

Please note that the exact position of operation's start point may be different from your selected point. The program will choose the nearest point from a number of possible start positions.

Create a Profile Contouring Operation Between Two Planes

This task shows how to insert a 'Between Two Planes' Profile Contouring operation in the program.

To create the operation you must define:

- the Contouring mode as Between two planes
- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths) .

Open the PrismaticMilling01.CATPart document, then select the desired NC Manufacturing workbench from the Start menu. Make the Manufacturing Program current in the specification tree.

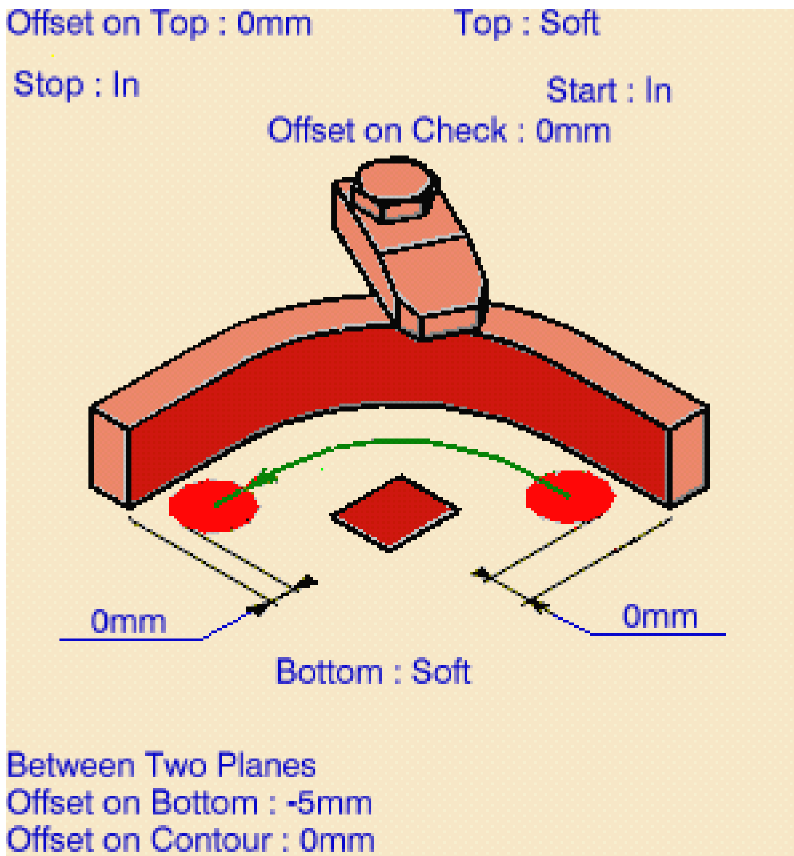
1. Select the Profile Contouring icon .

The Profile Contouring dialog box appears directly at the Geometry tab page .

This page includes a sensitive icon to help you specify the geometry to be machined.

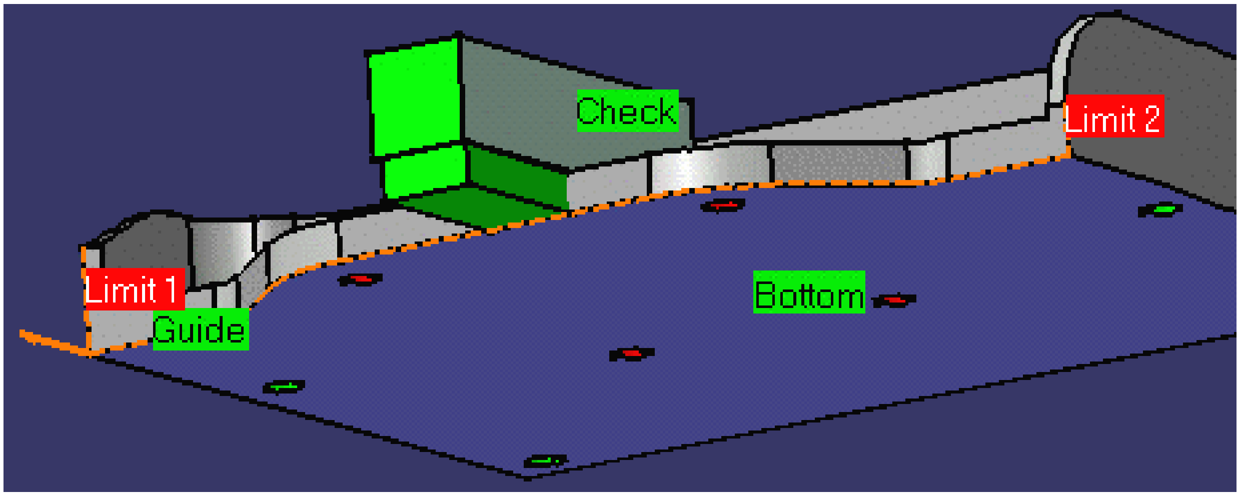
Right click the Contouring mode text and select **Between Two Planes**.

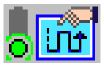
The part bottom and flanks in the icon are colored red indicating that this geometry is required for defining the operation.
All other geometry is optional.



2. Click the red bottom in the icon, then select the underside of the part in the 3D window.
3. Set the Bottom type to Soft by clicking the text, then set the Offset on Bottom to -5mm.
4. Click the red flank in the icon, then select the profile along the front edge of the part in the 3D window.
5. Click the first relimiting element in the icon, then select the horizontal edge at one end of the contour profile in the 3D window.
6. Click the second relimiting element in the icon, then select the horizontal edge at the other end of the contour profile in the 3D window.
7. Click the check element in the icon, then select the top face of the green fixture in the 3D window.

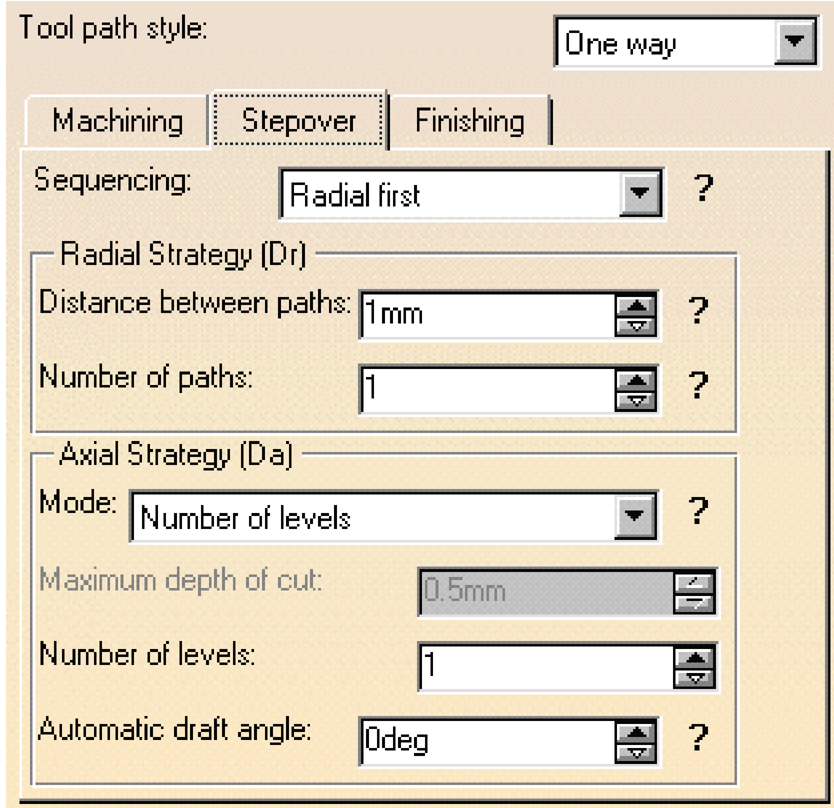
The bottom, guide, limit and check elements of the icon are now colored green indicating that this geometry is now defined. These are also indicated on the part.



8. Select the Strategy tab page  and choose the desired tool path style.

   You can then use the tab pages to set parameters for:

   - machining such as machining tolerance
   - stepover (see example)
   - finishing.

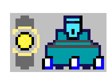


You can choose between the standard tip output and a cutter profile output by means of the **Output type** option in the Machining tab page.
If a cutter profile style is selected, both the tip and cutter profile will be visualized during tool path replay.

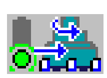
For cutter profile, cutter compensation instructions are generated in the NC data output. In this case, an approach macro must be defined to allow the compensation to be applied.

For macro types other than approach and retract, the compensation is not applied: these macros will be framed with CUTCOM/OFF and CUTCOM /left/right instructions.

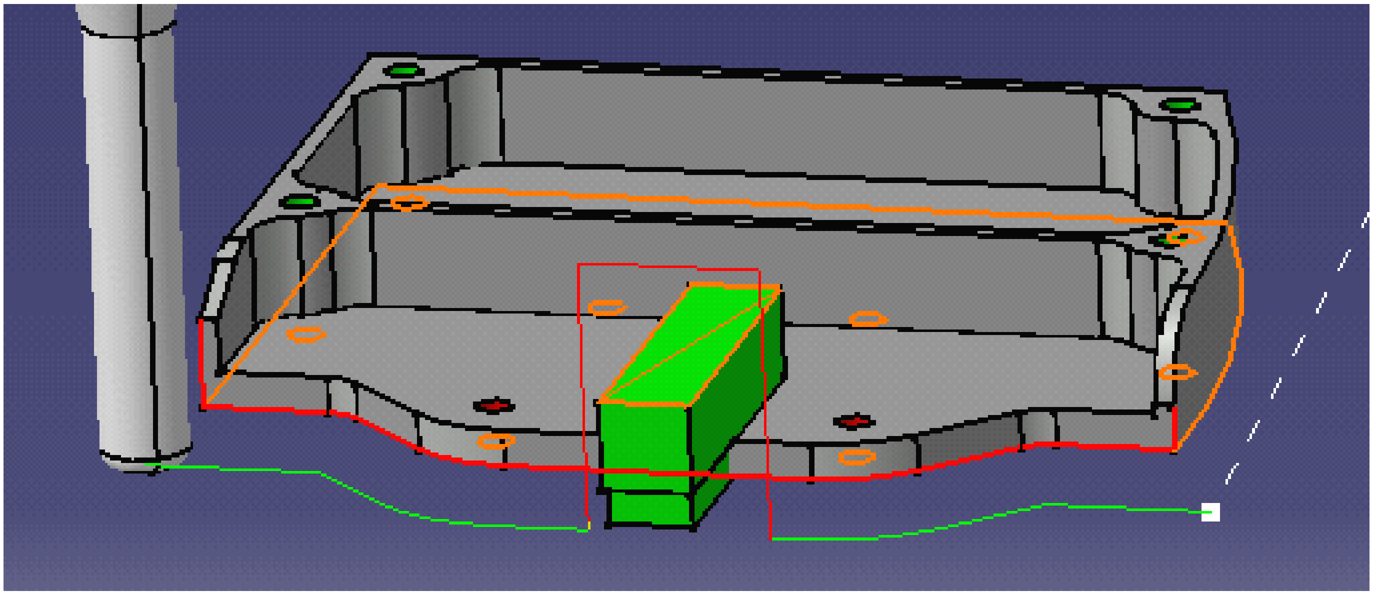
Any user defined PP words in macros are added to the cutter compensation instructions generated in the NC data output. Therefore you should be careful when specifying CUTCOM instructions in macros.

A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

9. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

10. Check the validity of the operation by replaying the tool path.



The specified operation uses a default linking macro to avoid collision with the selected fixture.

You can optimize the linking macro and add approach and retract macros to the operation in the Macros tab page. This is described in Define Macros of a Milling Operation.

11. Click OK to create the operation.

Create a Profile Contouring Operation Between Two Curves

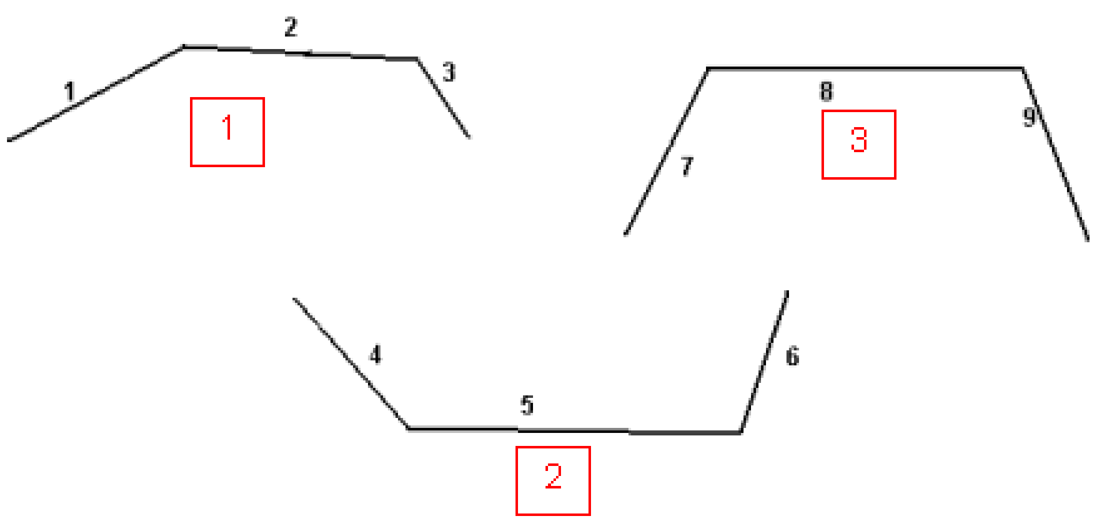
This task shows how to insert a 'Between Two Curves' Profile Contouring operation in the program.

To create the operation you must define:

- the Contouring mode as Between Two Curves
- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

This task also illustrates the capability to machine a discontinuous guiding curve.

You can machine several discontinuous groups of guiding elements in all Profile Contouring modes (except By Flank Contouring).



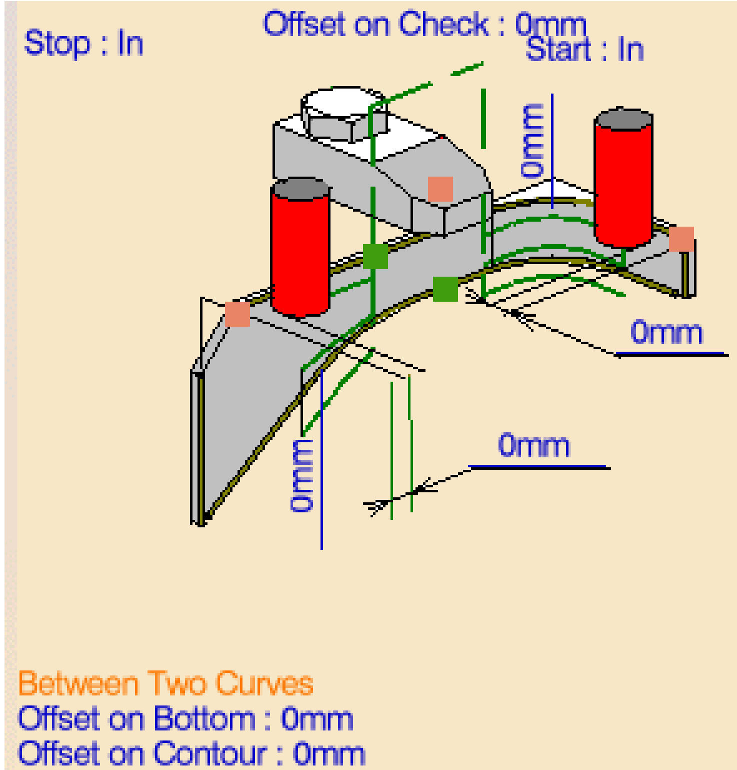
The order of selection of the geometric elements determines the order in which they will be machined.

Open the PrismaticMilling02.CATPart document, then select the desired NC Manufacturing workbench from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Profile Contouring icon.

   The Profile Contouring dialog box appears directly at the Geometry tab page.
   This page includes a sensitive icon to help you specify the geometry to be machined.

   Right click the Contouring mode text and select **Between Two Curves**.



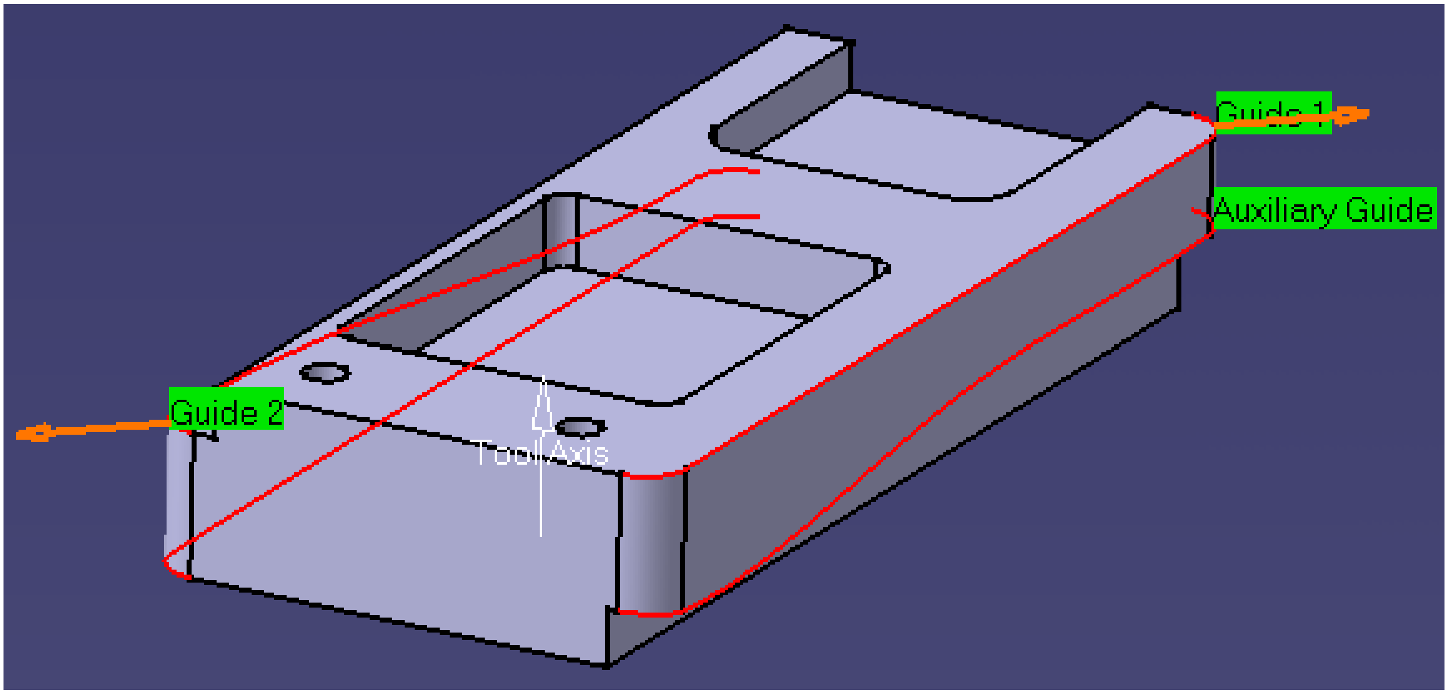
The top guiding curve in the icon is colored red indicating that this geometry is required for defining the operation.
All other geometry is optional.

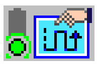
2. Click the top guiding curve in the icon, then in the 3D window:
   - select the three continuous edges on the top of the part as shown (Guide 1 in figure below)
   - select the three continuous edges of the downward slope on the other side of the part as shown (Guide 2 in figure below).

   During the selection, answer No to the question about inserting a line.
3. Click the bottom guiding curve in the icon, then in the 3D window:
   - select the three continuous edges of the downward slope the part as shown (Auxiliary Guide in figure below)
   - select the three continuous bottom edges on the other side of the part as shown.

   During the selection, answer No to the question about inserting a line.
4. If needed, set offsets on the geometric elements.

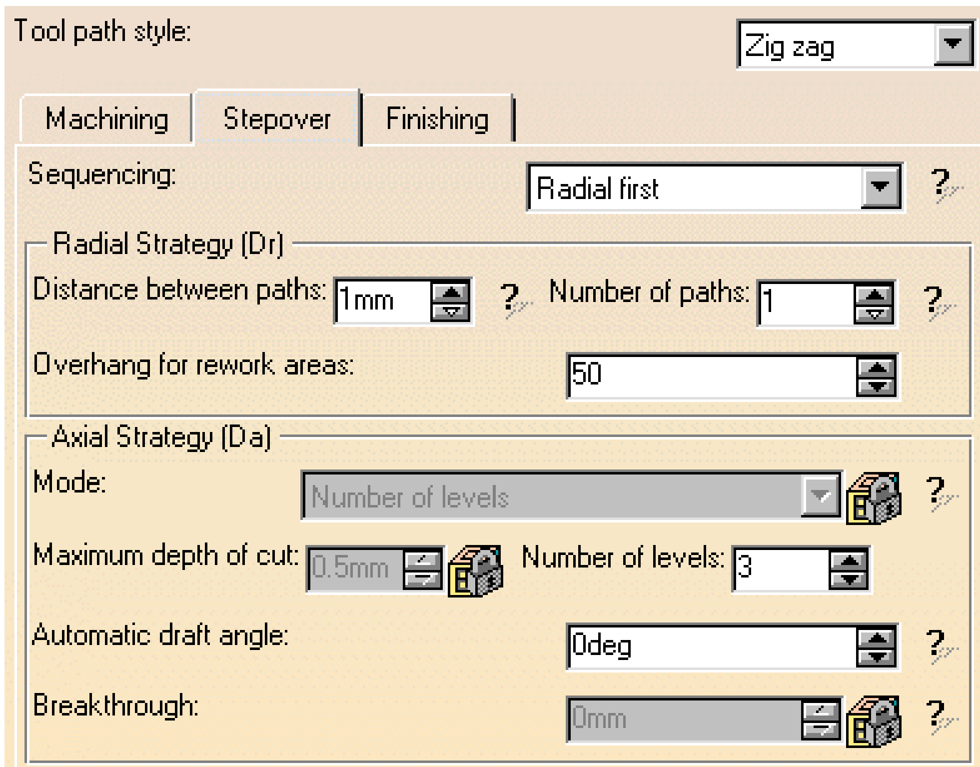
The guide and limit elements of the icon are now colored green indicating that this geometry is now defined. These are also indicated on the part.



5. Select the Strategy tab page  and choose the desired tool path style.

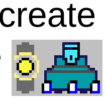
   You can then use the tab pages to set parameters for:

   - Machining such as machining tolerance
   - Stepover: in this example just set the Number of levels = 3.
   - Finishing.

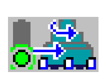


6. In the Macros tab page  you should add an appropriate Linking macro that will allow the tool to retract and approach the discontinuous guiding curves.

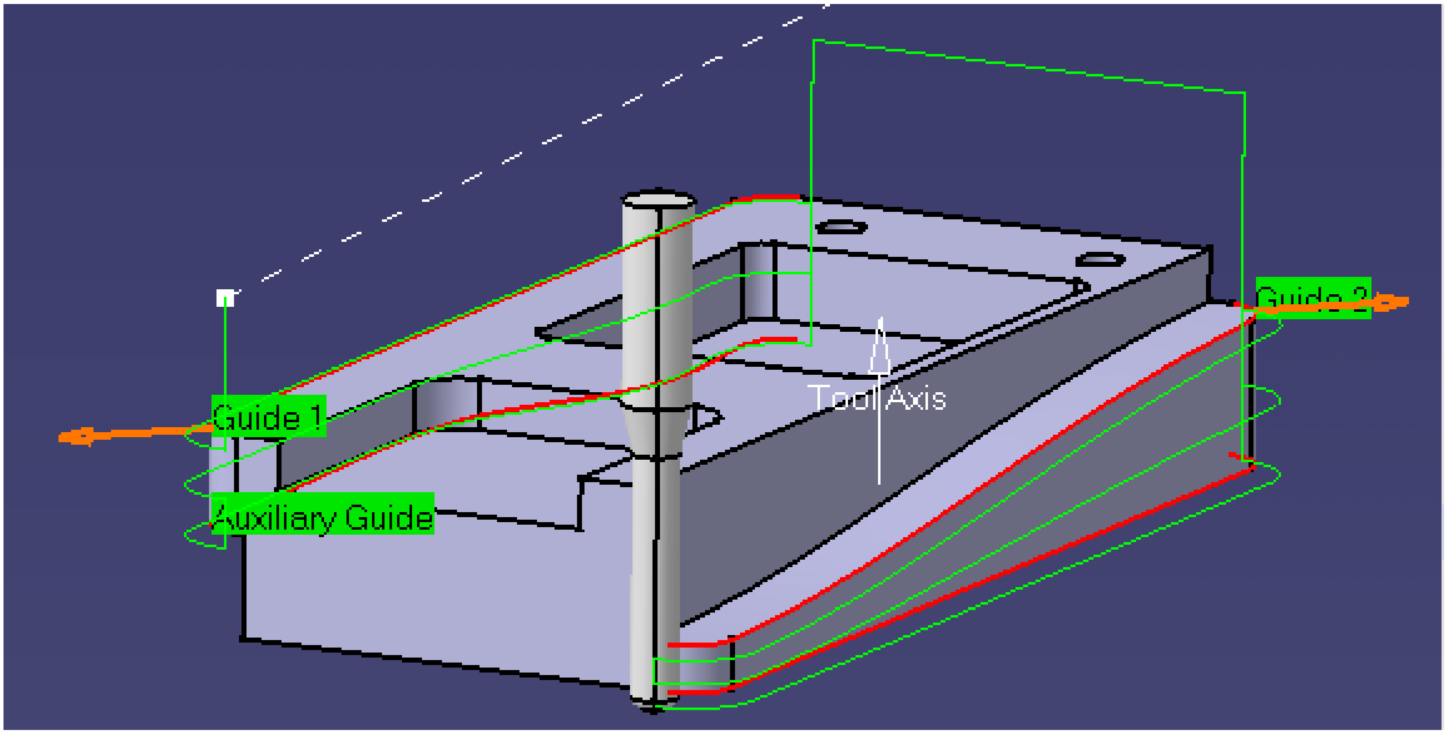
   This is described in Define Macros of an Operation.

A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

This procedure for this is described in Edit the Tool of an Operation.

7. If needed, select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

8. Check the validity of the operation by replaying the tool path.



Please note that the tool tip is shifted below the guiding curves by a distance equal to the tool corner radius. If you want the tool tip to exactly follow the guiding curves you should enter an appropriate Offset on Contour value.

9. Click OK to create the operation.

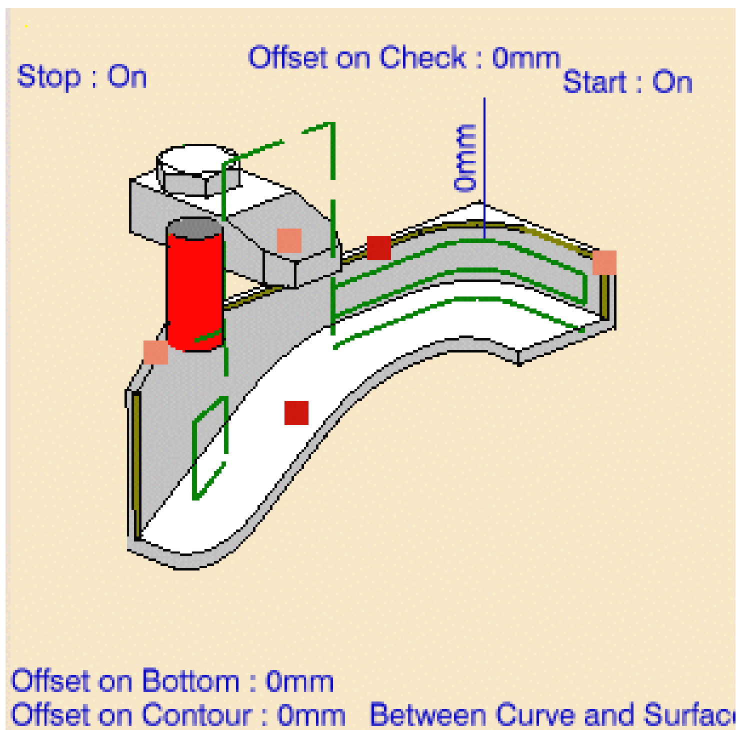
Create a Profile Contouring Operation Between a Curve and Surfaces

This task shows how to insert a 'Between Curve and Surfaces' Profile Contouring operation in the program.

To create the operation you must define:

- the Contouring mode as Between Curve and Surfaces
- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths) .

Open the PrismaticMilling02.CATPart document, then select the desired NC Manufacturing workbench from the Start menu. Make the Manufacturing Program current in the specification tree.



1. Select the Profile Contouring icon .

   The Profile Contouring dialog box appears directly at the Geometry tab page .

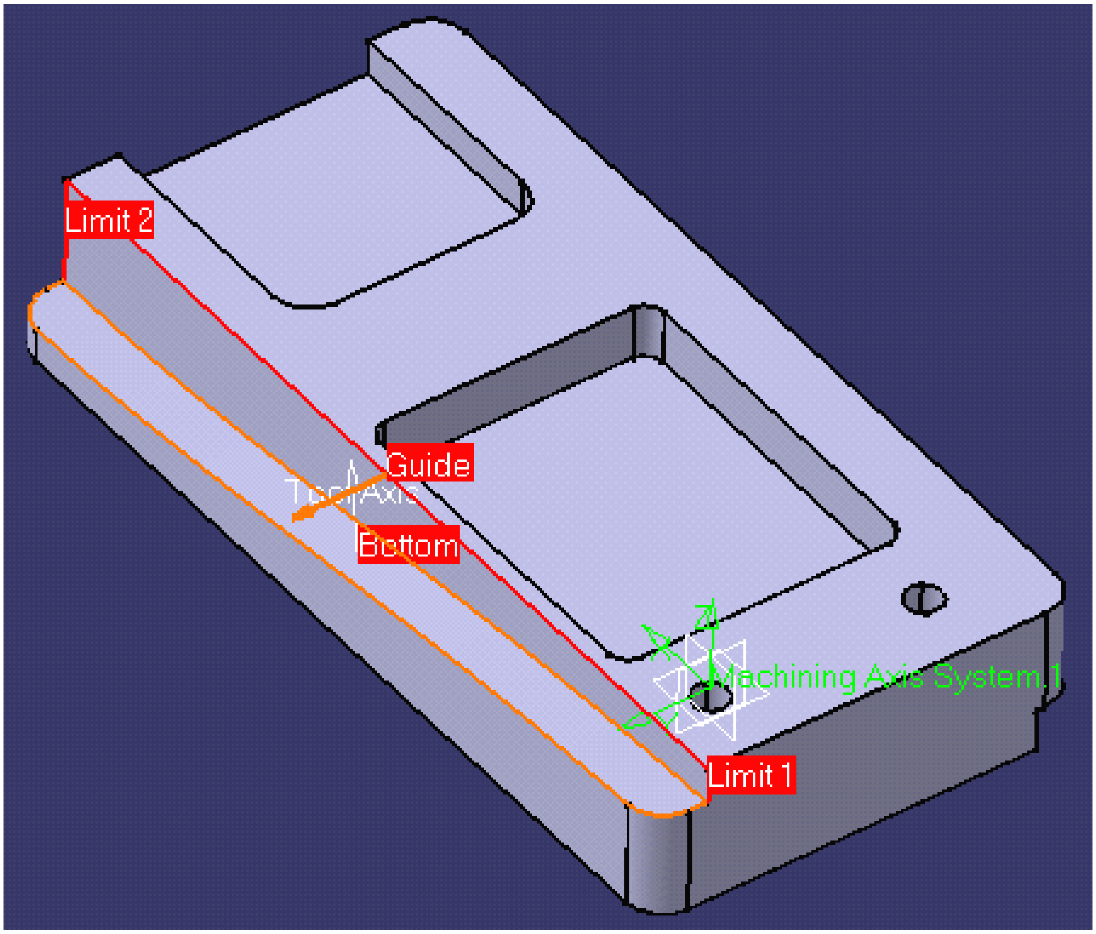
   This page includes a sensitive icon to help you specify the geometry to be machined.

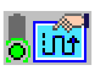
   Right click the Contouring mode text and select **Between Curve and Surfaces**.

   The top guiding curve and part bottom in the icon are colored red indicating that this geometry is required for defining the operation.
   All other geometry is optional.
2. Click the red bottom in the icon, then select the bottom surface of the part in the 3D window.
3. Click the top guiding curve in the icon, then select the top edge of the part in the 3D window.

4. Click the first relimiting element in the icon, then select a vertical edge at one end of the part in the 3D window.
5. Click the second relimiting element in the icon, then select the vertical edge at the other end of the part in the 3D window.
6. If needed, set offsets on the geometric elements.

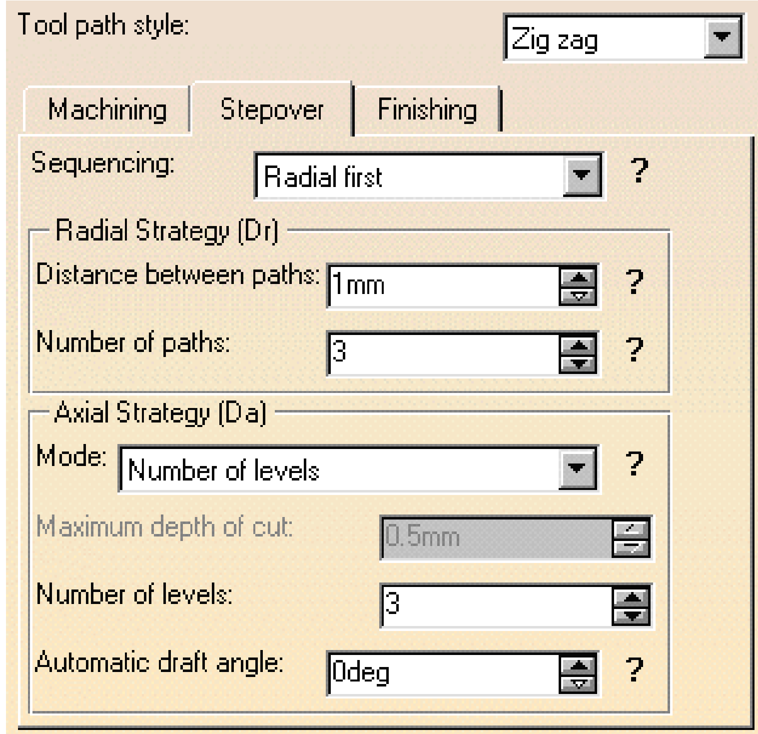
The guide and limit elements of the icon are now colored green indicating that this geometry is now defined. These are also indicated on the part.



7. Select the Strategy tab page  and choose the desired tool path style.

   You can then use the tab pages to set parameters for:

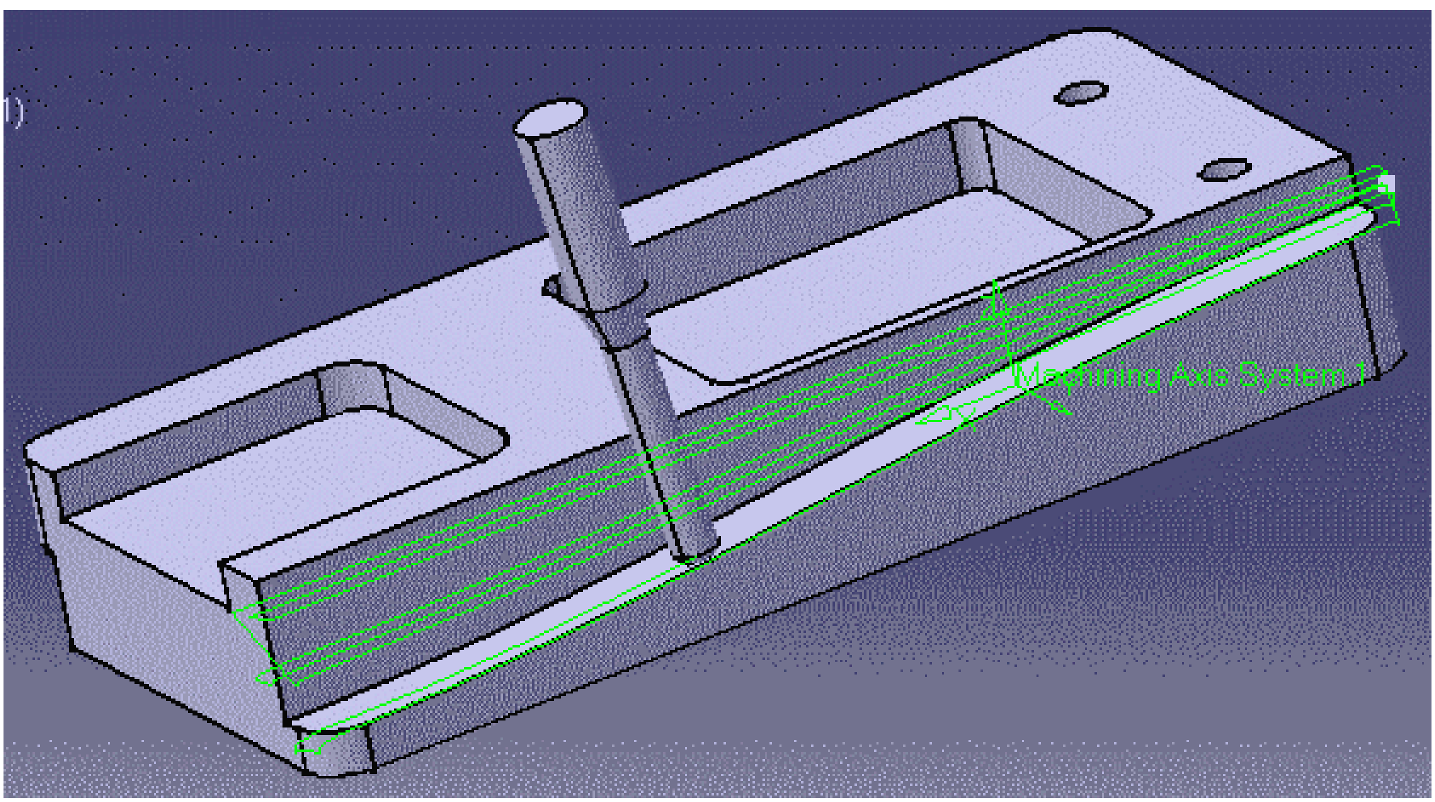
   - machining such as machining tolerance
   - stepover (see example)
   - finishing.



A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

8. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.
9. Check the validity of the operation by replaying the tool path.



You can add approach and retract motions to the operation in the Macros tab page . This is described in Define Macros of an Operation.

10. Click OK to create the operation.

Create a Profile Contouring Operation for Flank Contouring

This task shows how to insert a 'Flank Contouring' Profile Contouring operation in the program.

To create the operation you must define:

- the Contouring mode as Flank Contouring
- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths) .

Open the PrismaticMilling02.CATPart document, then select the desired NC Manufacturing workbench from the Start menu. Make the Manufacturing Program current in the specification tree.

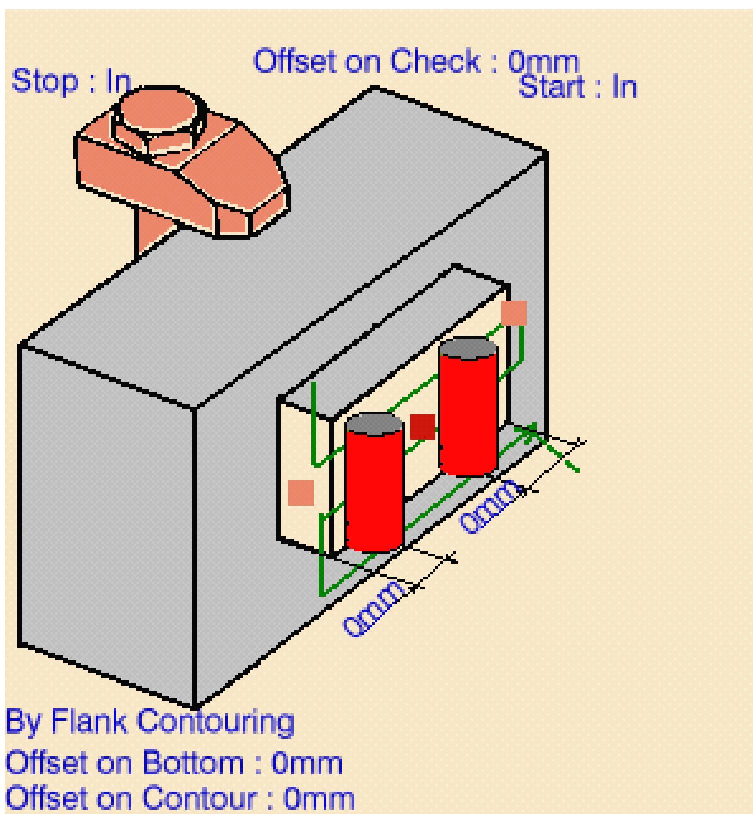
1. Select the Profile Contouring icon .

   The Profile Contouring dialog box appears directly at the Geometry tab page .

   This page includes a sensitive icon to help you specify the geometry to be machined.

   Right click the Contouring mode text and select **By Flank Contouring**.

   The guiding element in the icon is colored red indicating that this geometry is required for defining the operation. All other geometry is optional.



2. Click the guiding element in the icon, then select the vertical face of the part in the 3D window.
3. Click the first relimiting element in the icon, then select a vertical edge at one end of the part in the 3D window.

4. Click the second relimiting element in the icon, then select the vertical edge at the other end of the part in the 3D window.

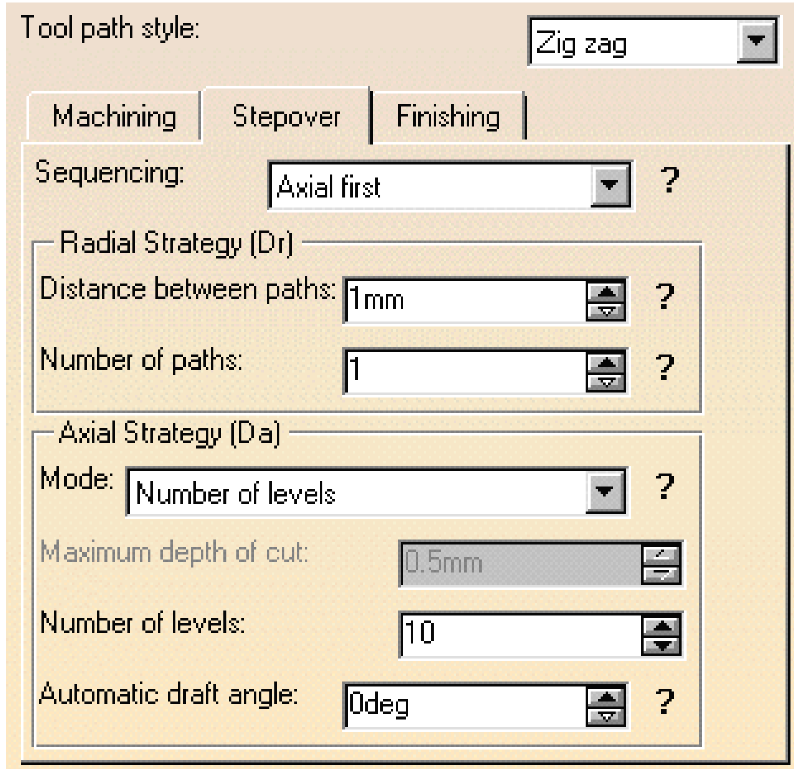
The guide and limit elements of the icon are now colored green indicating that this geometry is now defined. These are also indicated on the part.

5. If needed, set offsets on the geometric elements.

6. Select the Strategy tab page and choose the desired tool path style.

You can then use the tab pages to set parameters for:

- machining such as machining tolerance
- stepover (see example)
- finishing.

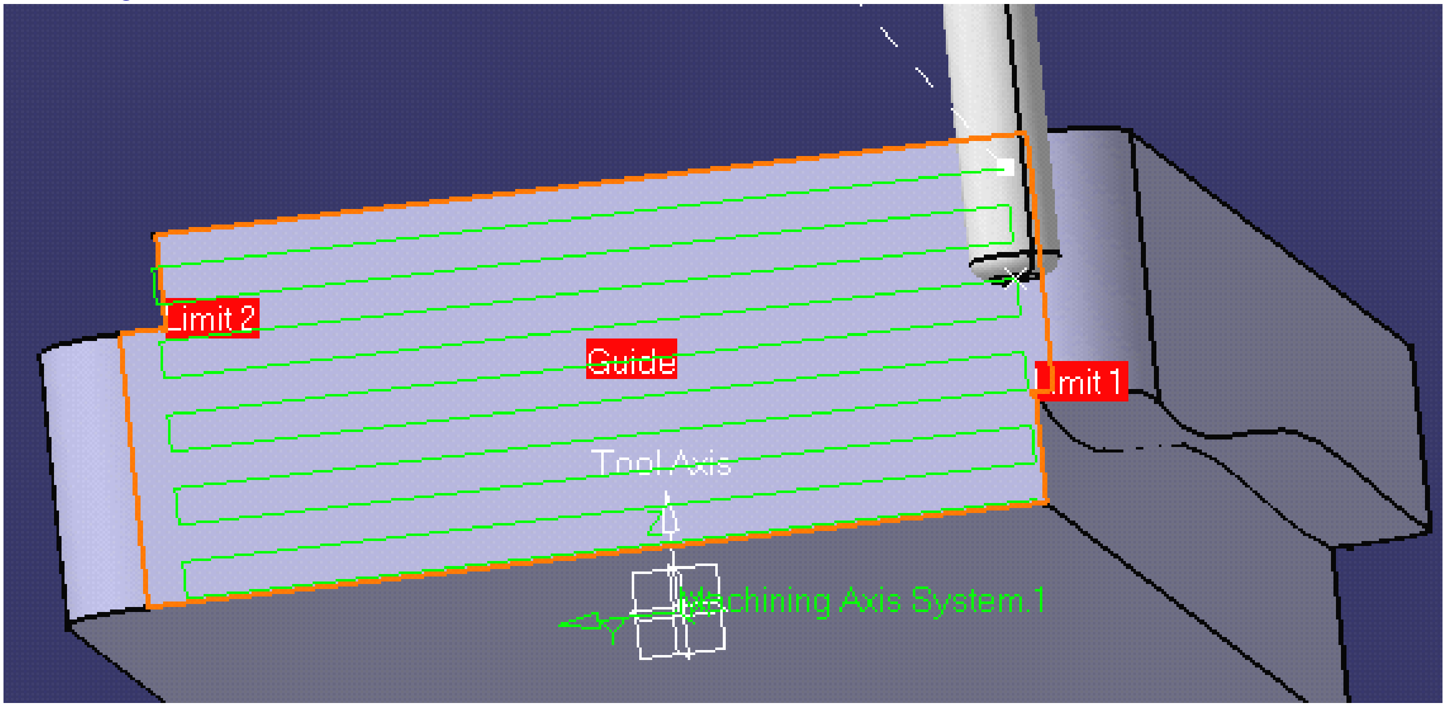


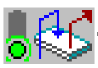
A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

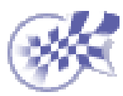
7. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

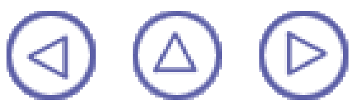
8. Check the validity of the operation by replaying the tool path.



You can add approach and retract motions to the operation in the Macros tab page . This is described in Define Macros of an Operation.

9. Click OK to create the operation.





Create a Point to Point Operation

This task shows how to insert a Point to Point operation in the program.

To create the operation you must define:

- a sequence of elementary tool motions and machining strategy parameters
- the tool that will be used
- the feedrates and spindle speeds
- the macros (transition paths).

Open the PrismaticMilling02.CATPart document, then select NC Manufacturing > Prismatic Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Point to Point icon.

   A Point to Point entity along with a default tool is added to the program.

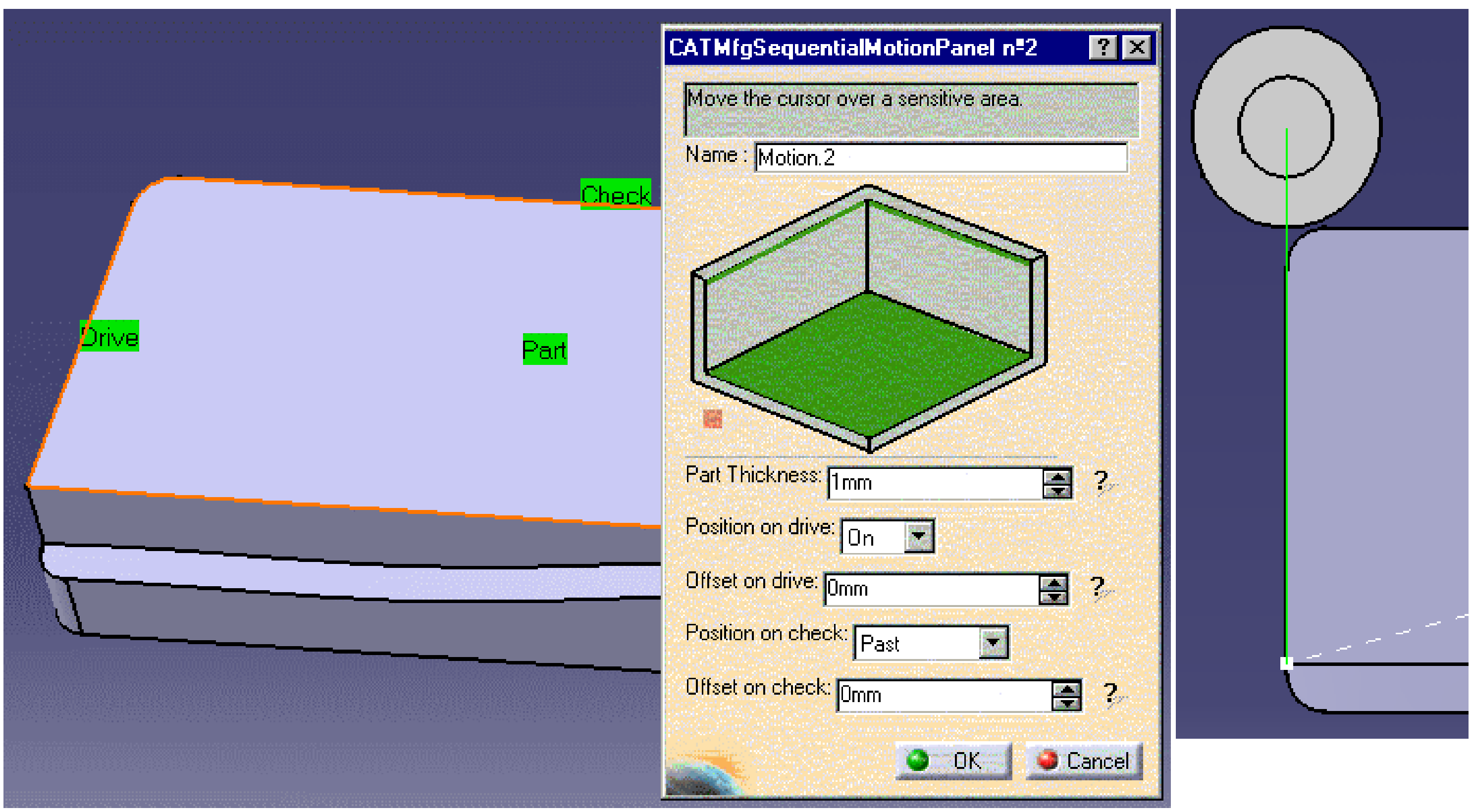
   The Point to Point dialog box appears directly at the Strategy tab page.

   This Motions tab allows you to define the elementary Goto Point and Goto Position motions making up the machining operation.

2. Click the Goto Point icon, then select a corner point on the underside of the part.

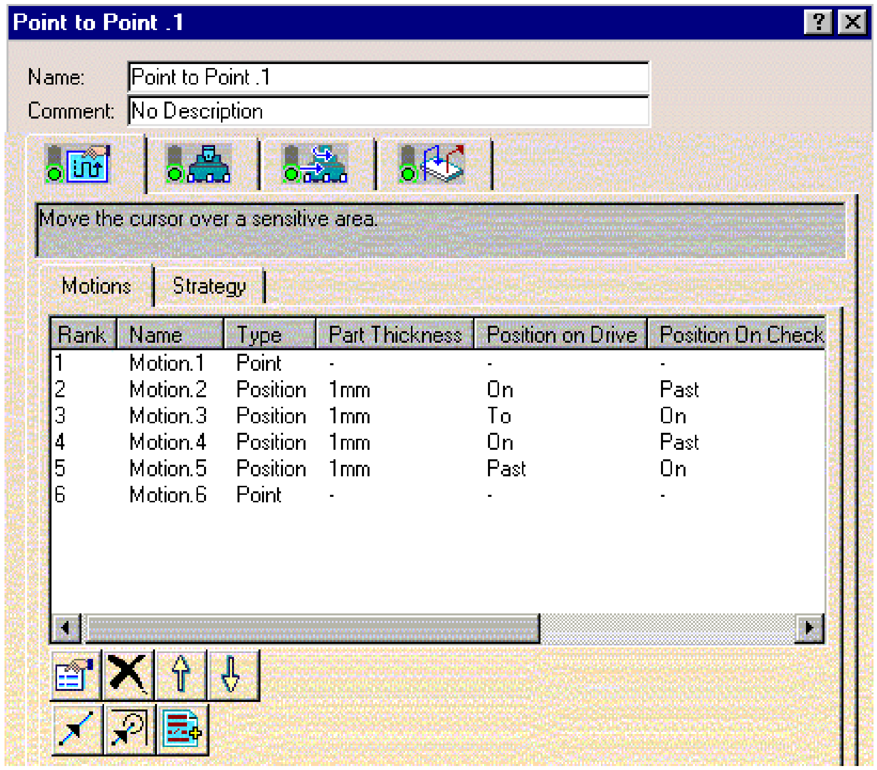
   Just double-click to end point selection. The first tool motion is defined and appears in the list in the dialog box.

3. Click the Goto Position icon. A dialog box appears to help you specify the part, drive and check elements as well as positioning conditions (To / On / Past).



4. Just click OK when you have specified the desired elements and conditions. The second tool motion is defined and appears in the list in the dialog box.

5. Add other Goto Point and Goto Position motions as shown in the figure below.



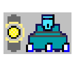
In this dialog box you can:

- add PP words to the list by clicking on the PP words icon and specifying the desired syntax.
- move motions up or down the list by the Arrow icons or remove motions by means of the Remove icon.
- edit the properties of a motion by clicking the Properties icon.

6. Select the Strategy tab to specify machining parameters.

If needed, double click the sensitive text in the icon to specify an offset along the tool axis.

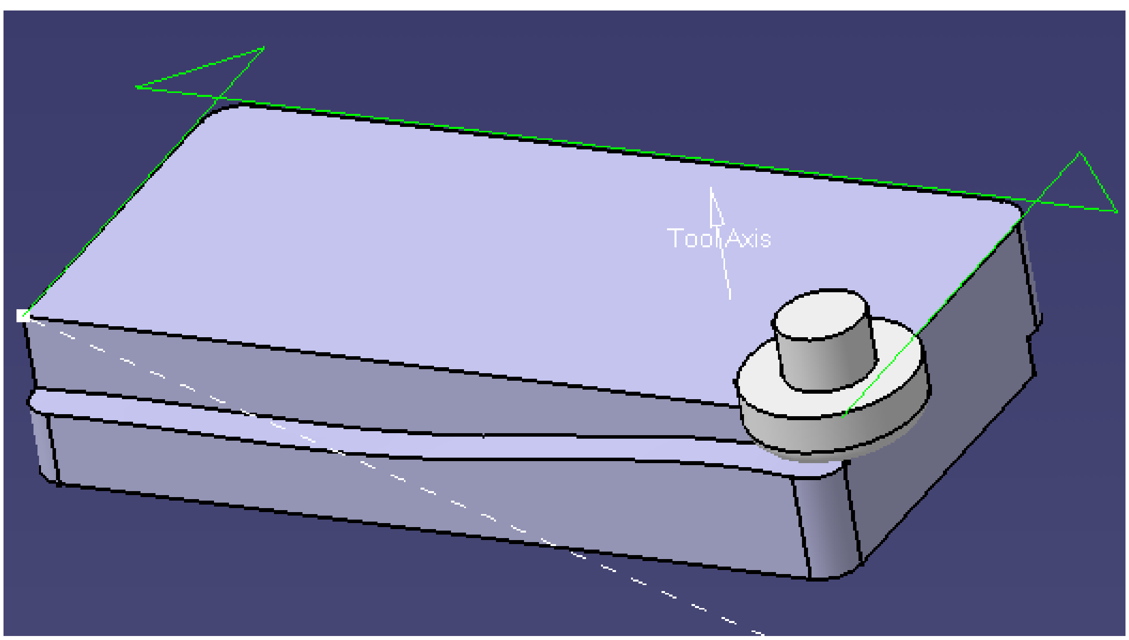
Define the tool axis direction by first selecting the axis representation in the sensitive icon then specifying the direction in the dialog box that appears.

7. Select the Tool tab page  to replace the default tool by a more suitable one.

Select the Face Mill icon. A 50mm diameter face mill is proposed. You can adjust the parameters as required.

See Edit the Tool of an Operation for more information about selecting tools.

Check the validity of the operation by replaying the tool path.



8. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.
9. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Operation.
10. Click OK to create the operation.

You can use the contextual menu commands to insert or remove points and to assign local feedrates.

By selecting a circle, its center is taken as the point to machine.

Points of an associated sketch can also be selected.

Create a Curve Following Operation

This task shows how to insert a **Curve Following** operation in the program.

To create the operation you must define:

- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

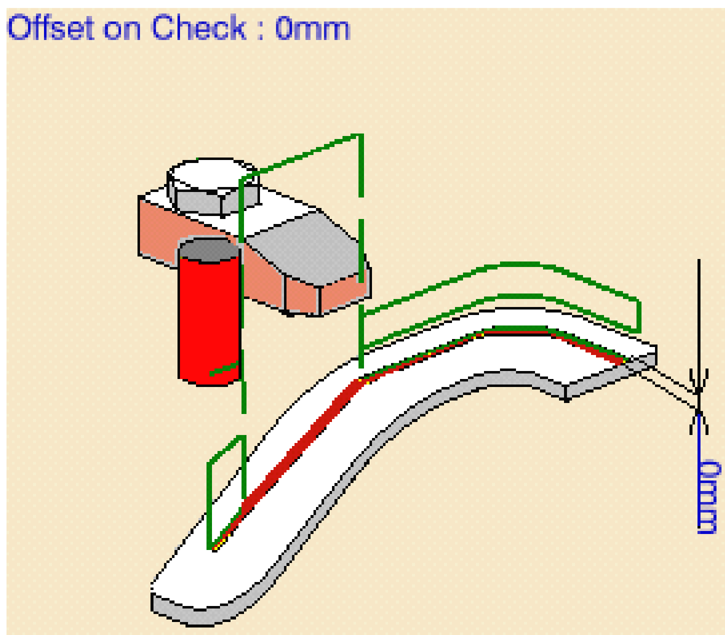
Open the PrismaticMilling02.CATPart document, then select the desired NC Manufacturing workbench from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Curve Following icon.

   The Curve Following dialog box appears directly at the Geometry tab page.

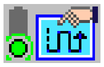
   This page includes a sensitive icon to help you specify the geometry to be machined.

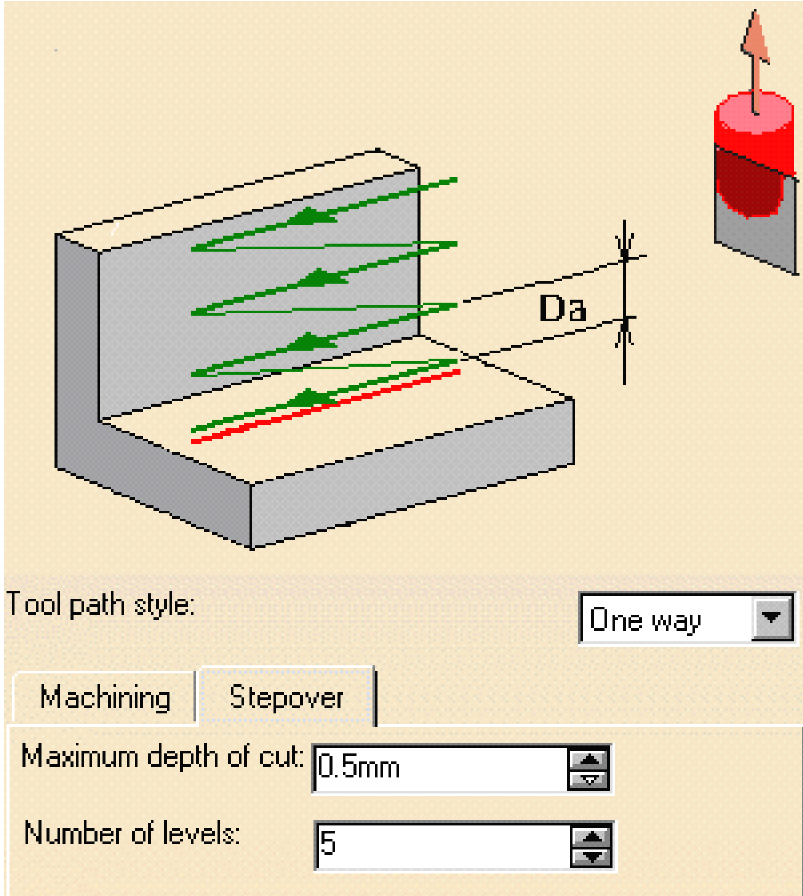
   The guiding curve in the icon is colored red indicating that this geometry is required for defining the operation. All other geometry is optional.

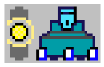


2. Click the guiding curve in the icon, then select the desired curve in the 3D window.
3. If needed, set an axial offset on the guiding curve.

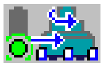
   The guide element of the icon is now colored green indicating that this geometry is now defined. It is also indicated on the part.

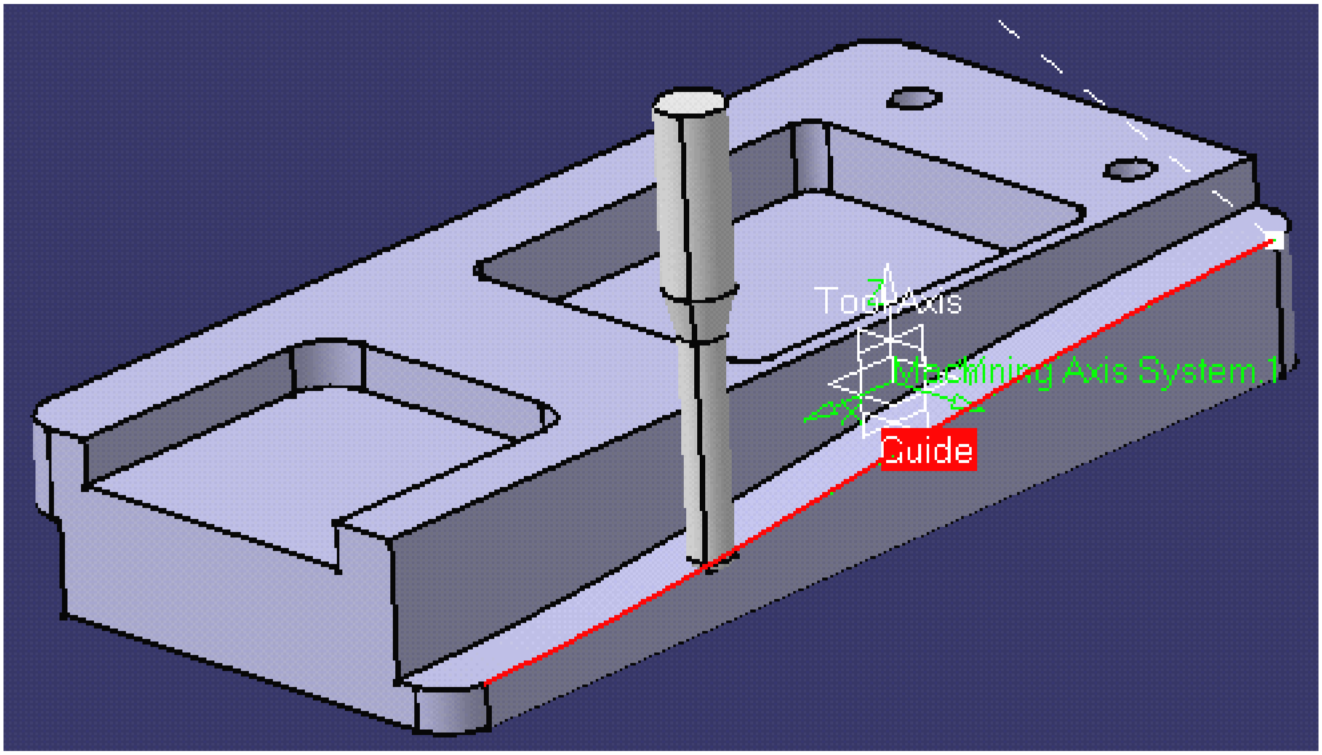
4. Select the Strategy tab page  to specify the main machining strategy parameters as shown.

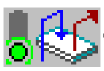


A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

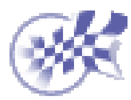
This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.
6. Check the validity of the operation by replaying the tool path.



You can add approach and retract motions to the operation in the Macros tab page . This is described in Define Macros of an Operation.

7. Click OK to create the operation.



Create Operations for Reworking Corners and Channels

This task shows how to create operations for reworking corners and channels by making use of **prismatic rework areas**. The areas to be reworked are identified by means of a Rework feature.

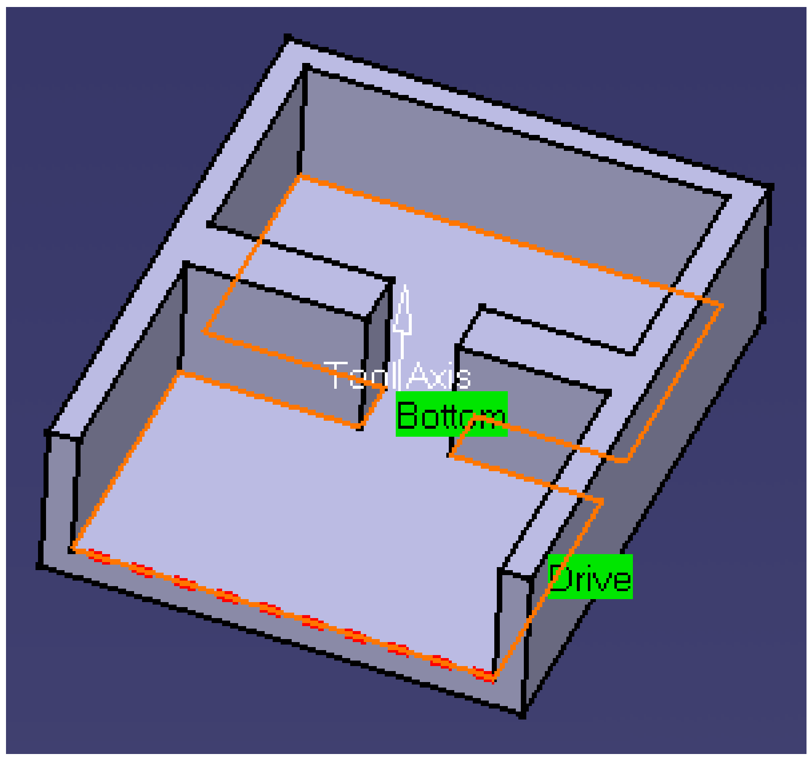
You will first create a Pocketing operation to rough cut the part. Then, you will see how corners and channels can be quickly identified and then reworked using different operations.

Open the Rework01.CATPart document, then select NC Manufacturing > Prismatic Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Pocketing icon .
2. Click the red Bottom in the icon then select the pocket bottom in the 3D window.
3. Select the Tool tab page to specify a tool with nominal diameter = 40mm.

   You can refer to Edit the Tool of an Operation for more information.
4. Select the Macros tab page to specify a Return in a Level macro with linear Approach and Retract paths of 50mm.

   You can refer to Define Macros of an Operation for more information.

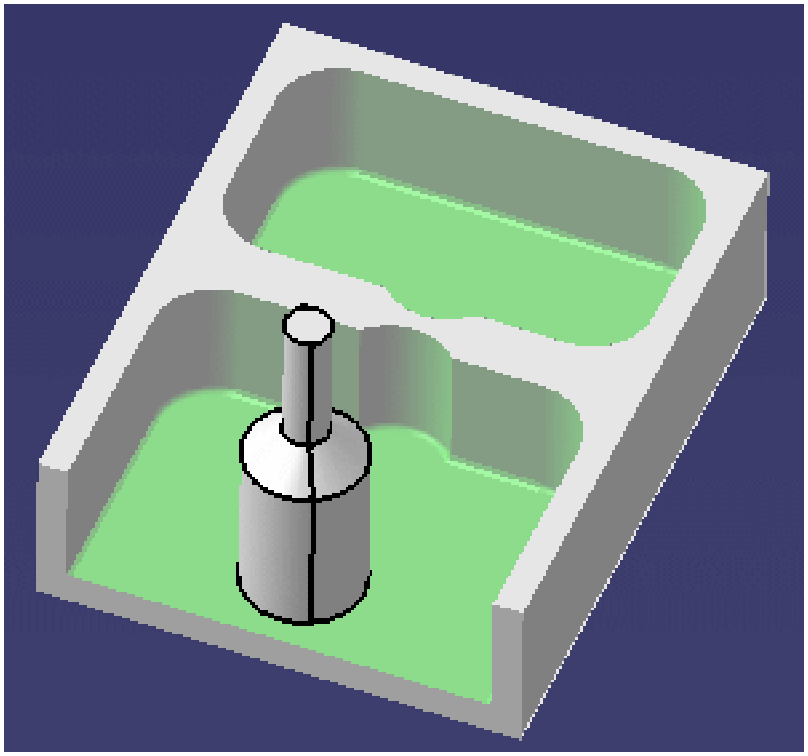


5. Check the Pocketing operation by replaying the tool path in Video mode.

   You can see that the pocket is rough cut by this operation.

   Close the Replay dialog box.

   Rename the operation Rough Pocketing then click OK to create it.

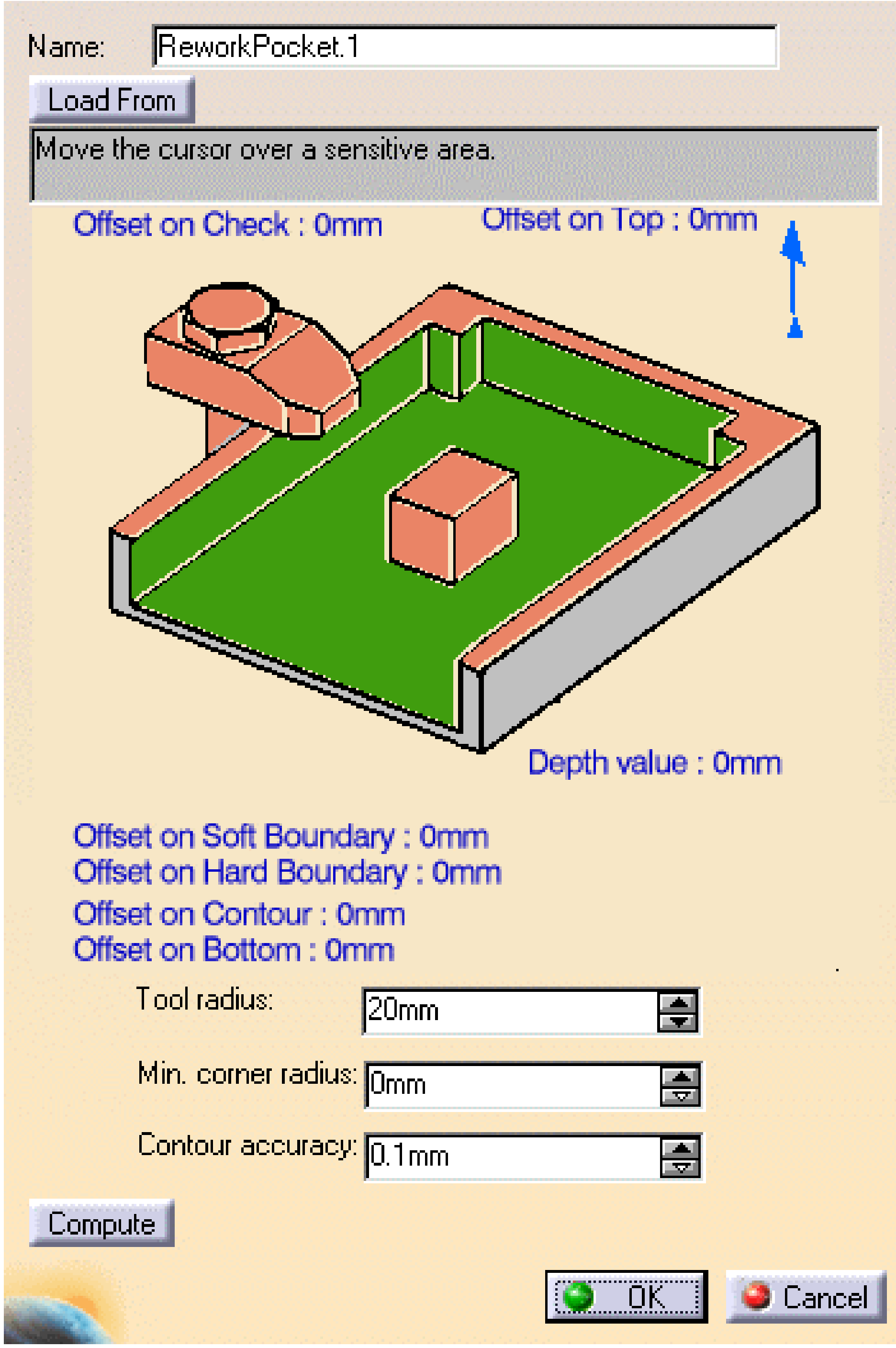


6. Select the Prismatic Rework Area icon .

   In the dialog box that appears, click **Load From** then select the Pocketing operation that you have just created.

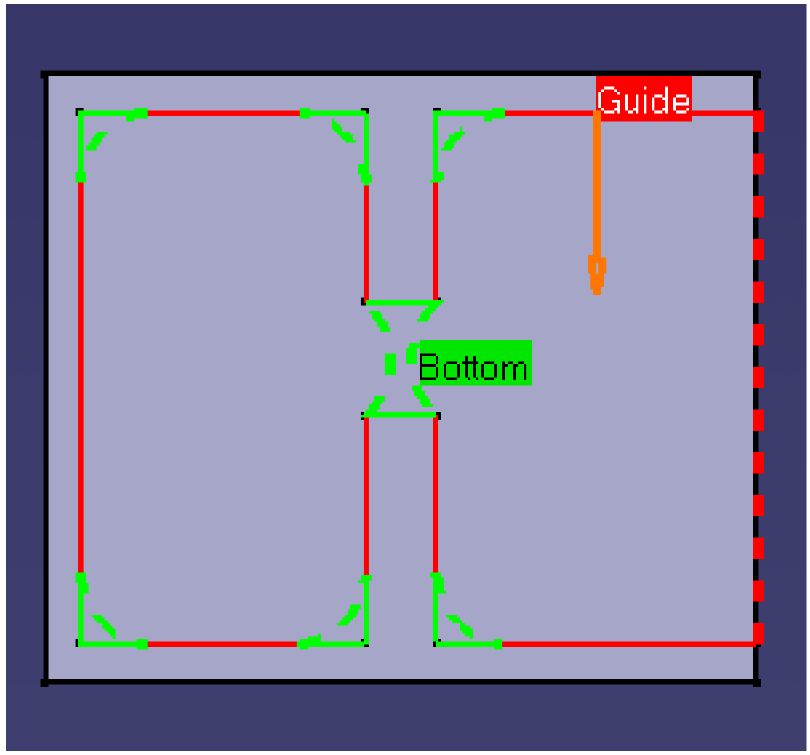
   The *Rework feature* is initialized with the geometry and other characteristics of the defined operation.

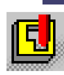
   You can rename this feature (for example, ReworkPocket.1).

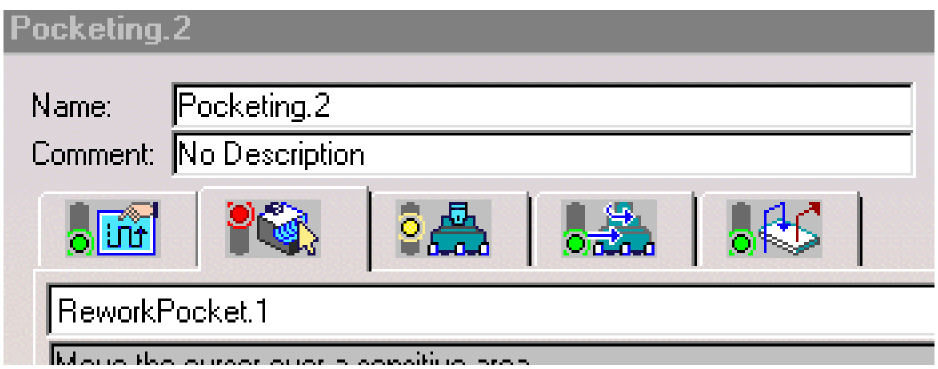


7. Click **Compute** to visualize the areas to rework.

   Here you can identify 6 corners and one channel to rework.

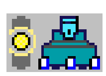


8. Select the Pocketing icon  to create an operation to machine the channel left by the first operation.
9. Select the ReworkPocket.1 feature using the combo in the Geometry tab page.



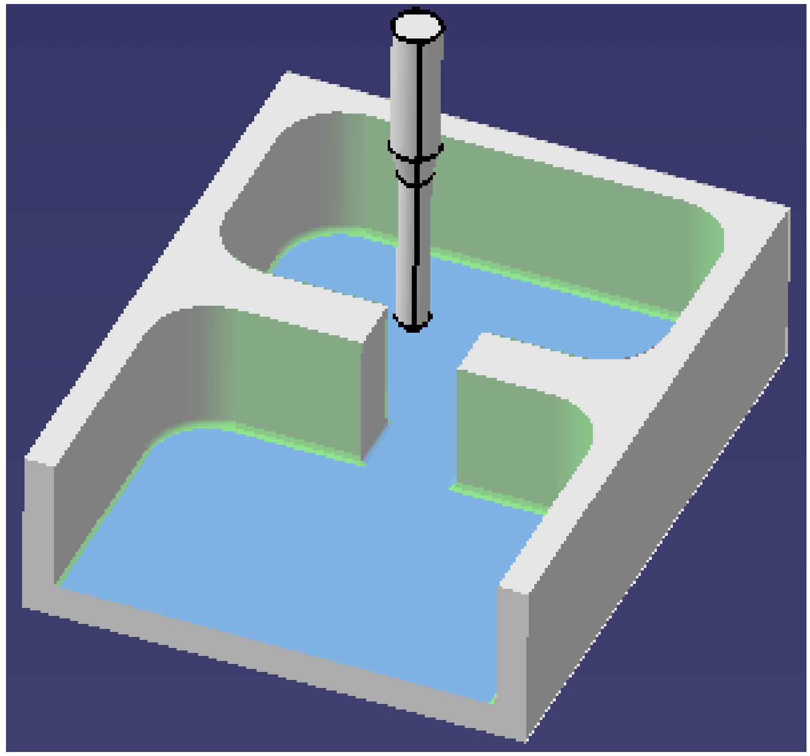
The dialog box is then updated with information from the feature.

Choose Channel from the Rework Area Type combo.

10. Select the Tool tab page  to specify a tool with nominal diameter = 10mm.
11. Select the Strategy tab page  and set the Overhang value to 100.
12. Select the Macros tab page  to specify:
    - an Approach macro with linear Approach path of 50mm
    - a Return in a Level macro with linear Approach and Retract paths of 50mm
    - a Retract macro with linear Retract path of 50mm.

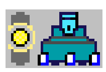
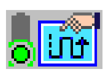
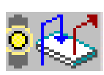
13. Check the operation by replaying the tool path in Video mode.

    You can see that the channel is machined by this operation.



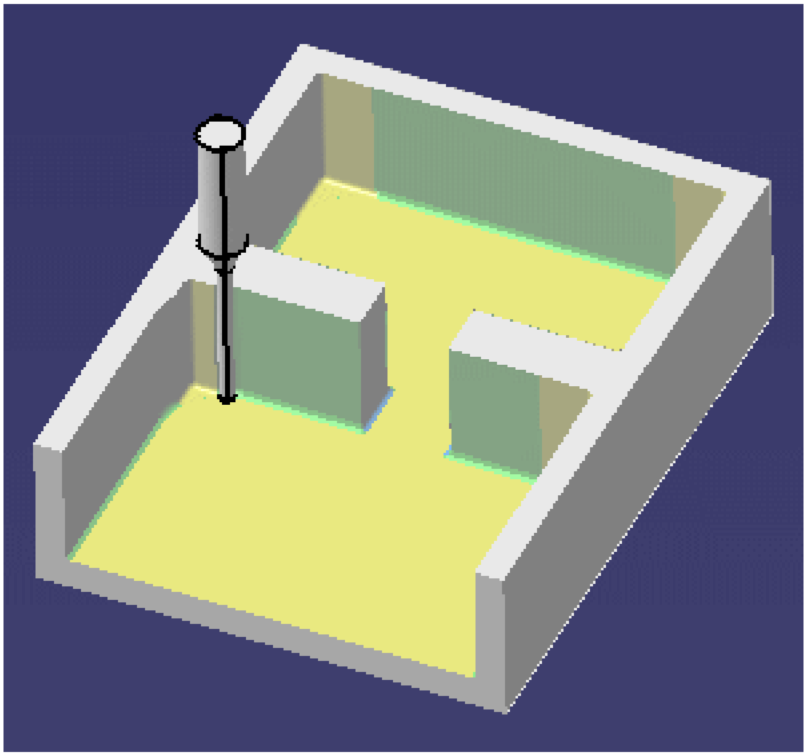
14. Rename the operation Channel Rework then click OK to create it.
15. Select the Profile Contouring icon  to create an operation to machine the corners left by the first operation.

    Select the ReworkPocket.1 feature using the combo in the Geometry tab page.

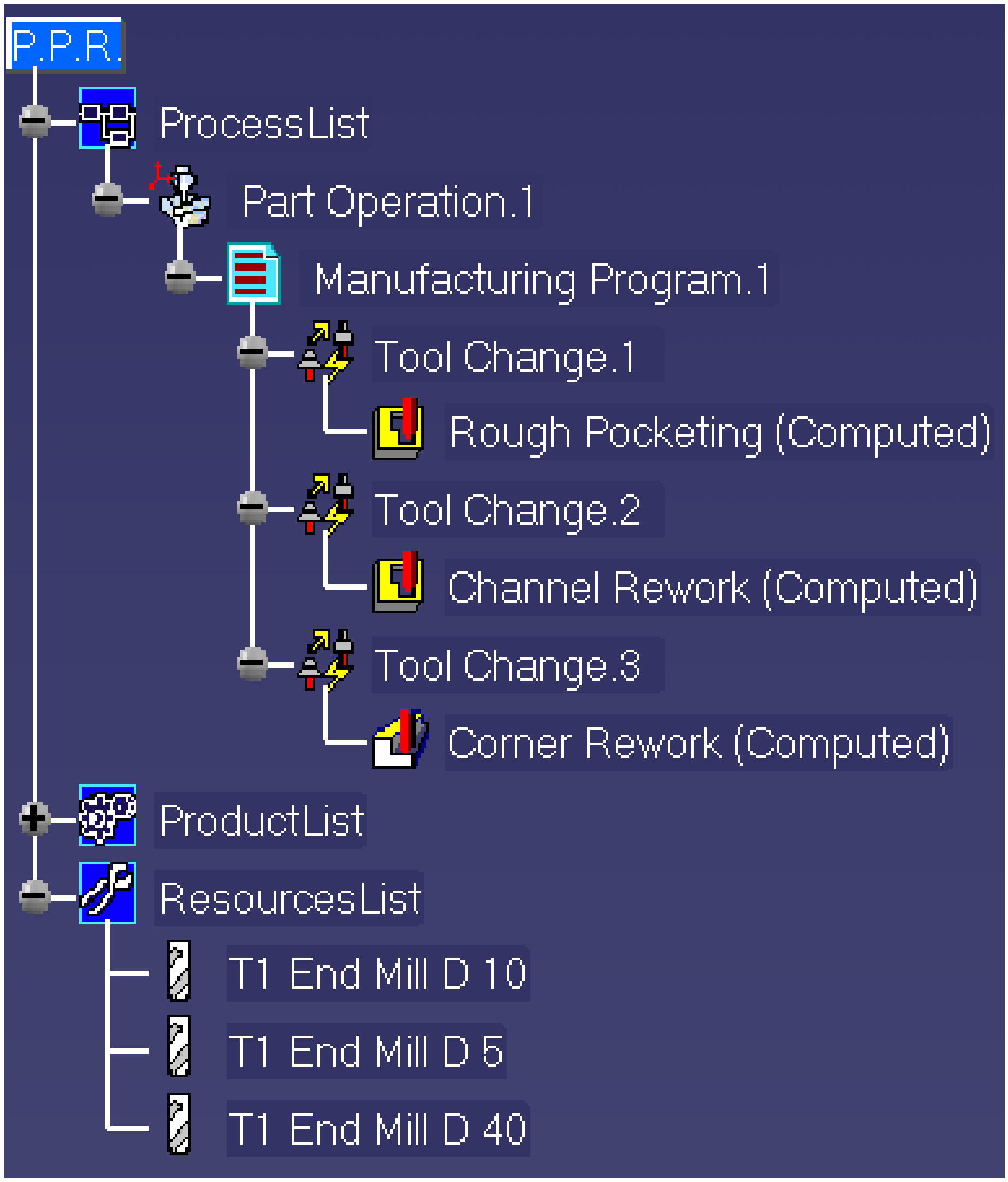
    The dialog box is then updated with information from the feature.
16. Select the Tool tab page  to specify a tool with nominal diameter = 5mm.
17. Select the Strategy tab page  and set:
    - Overhang for Rework Areas to 100
    - Distance between paths to 2mm.
18. Select the Macros tab page  to specify:
    - an Approach macro with linear Approach path of 50mm
    - a Return in a Level macro with linear Approach and Retract paths of 50mm
    - a Retract macro with linear Retract path of 50mm.

19. Check the operation by replaying the tool path in Video mode.

You can see that the corners are machined by this operation.



20. Rename the operation Corner Rework then click OK to create it.



This user task illustrates the **Load From** capability in which the Rework feature is initialized with the geometric and other characteristics of a machining operation.

You can also set the characteristics of the Rework feature manually by means of the dialog box.

This capability can be use with Pocketing and Profile Contouring operations.

2.5 to 5-axis Drilling Operations

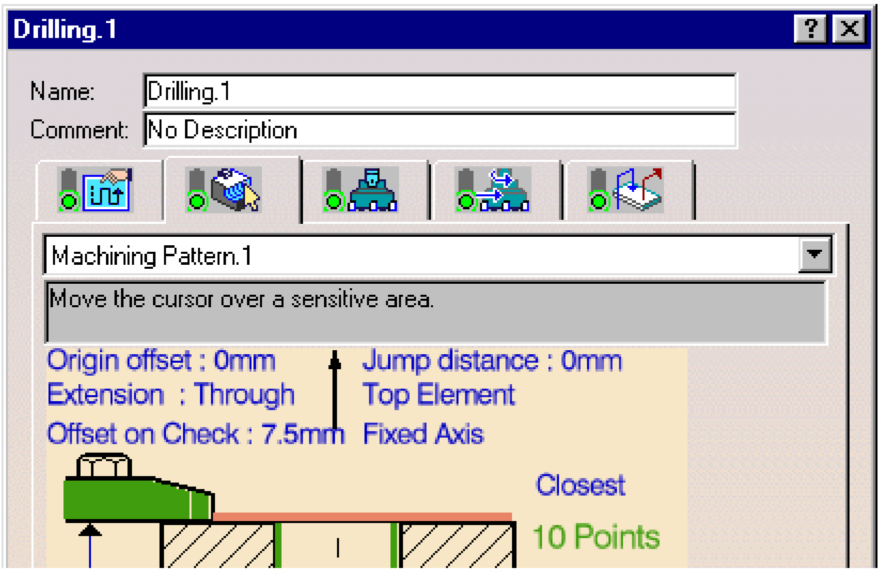
The drilling (or *axial machining*) operations described in this section are intended to cover the hole making activities in your NC manufacturing program. See Supported Axial Machining Operation Types.

The following paragraphs highlight some important information about these operations.

In particular, the commands and capabilities included in the Geometry tab page of the Axial Machining Operation dialog box allow support of multi-axis as well as fixed axis drilling.

Machining Patterns

In the Geometry tab page, the operation is assigned a machining pattern, which is initially empty. The identifier of this machining pattern appears in the combo at the top of this page. When you select hole points, these positions are added to the pattern.



You can assign a machining pattern to an axial machining operation by clicking the sensitive text (No Points or x Points). This opens the Pattern Selection dialog box that lists all available patterns. Just select one of the patterns and double click to exit.

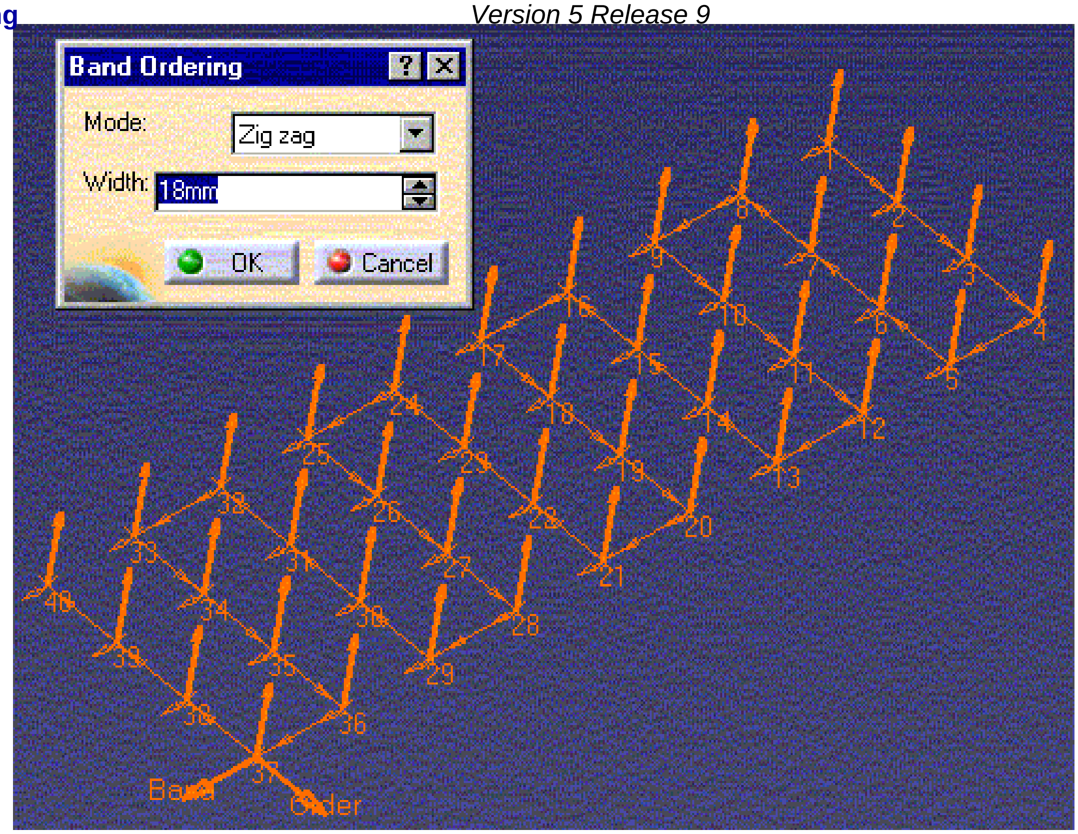
If there are already machining patterns on previous operations, the combo allows a quick selection of an existing pattern. This provides a shortcut to the selection method described above.

Ordering Strategies

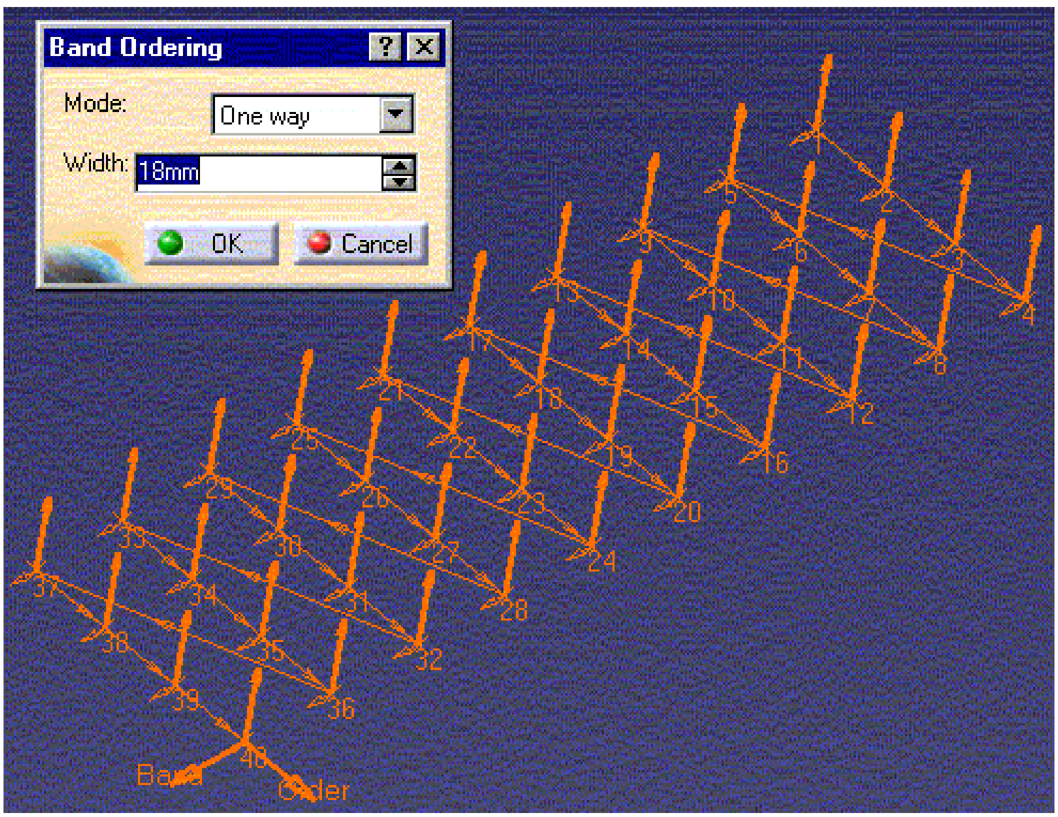
Holes that are selected for a machining pattern can be ordered according to the following modes:

- **Closest**: to obtain the shortest possible tool path
- **Manual**: to obtain a user-defined numbered sequence
- **By Band**: to obtain a Zig Zag or One Way configuration according to a set of bands that have a user-defined width.

  Zig Zag ordering of a pattern of 40 points for a band width of 18mm is illustrated below:



One Way ordering of the same points and same band width is illustrated below:



Overall Tool Axis Orientation

Right click the *Tool axis strategy* sensitive text in the Geometry tab page and select one of the following options to specify the general tool axis orientation:

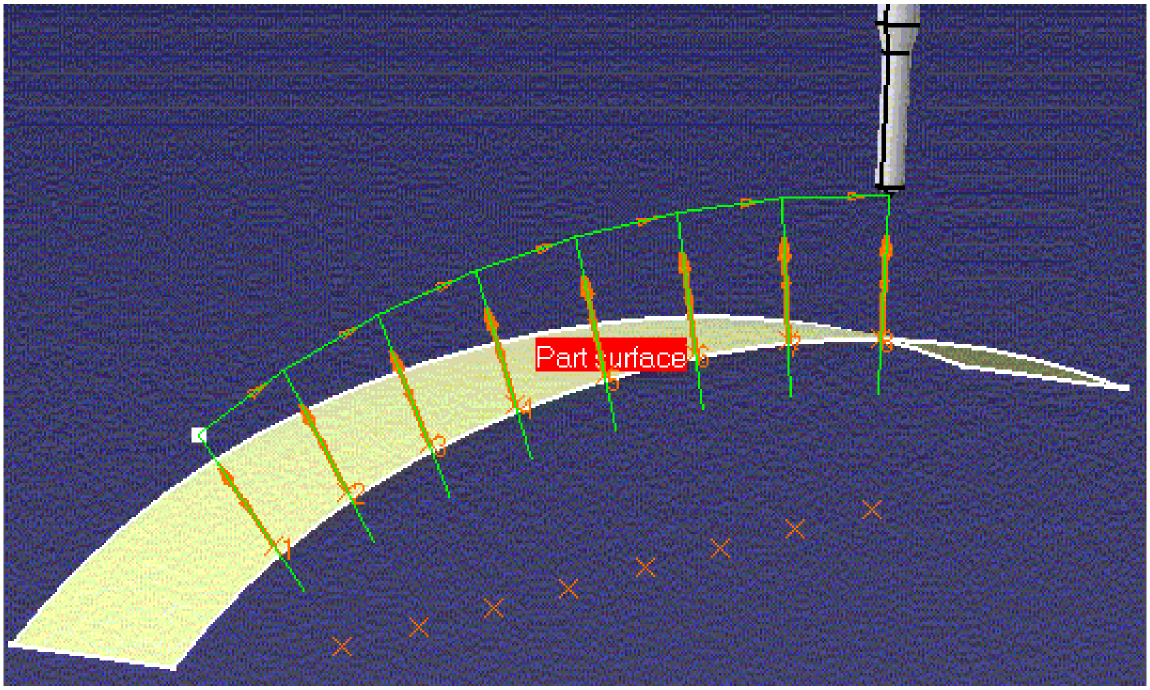
- **Fixed Axis**: the tool axis orientation is the same for all the selected points
- **Variable Axis**: the tool axis orientation can vary from one point to another
- **Normal to PS Axis**: the tool axis orientation is determined by the normal to the selected part surface.

Note that the tool axis orientation can be inverted by clicking the tool axis symbol in the Geometry tab page.

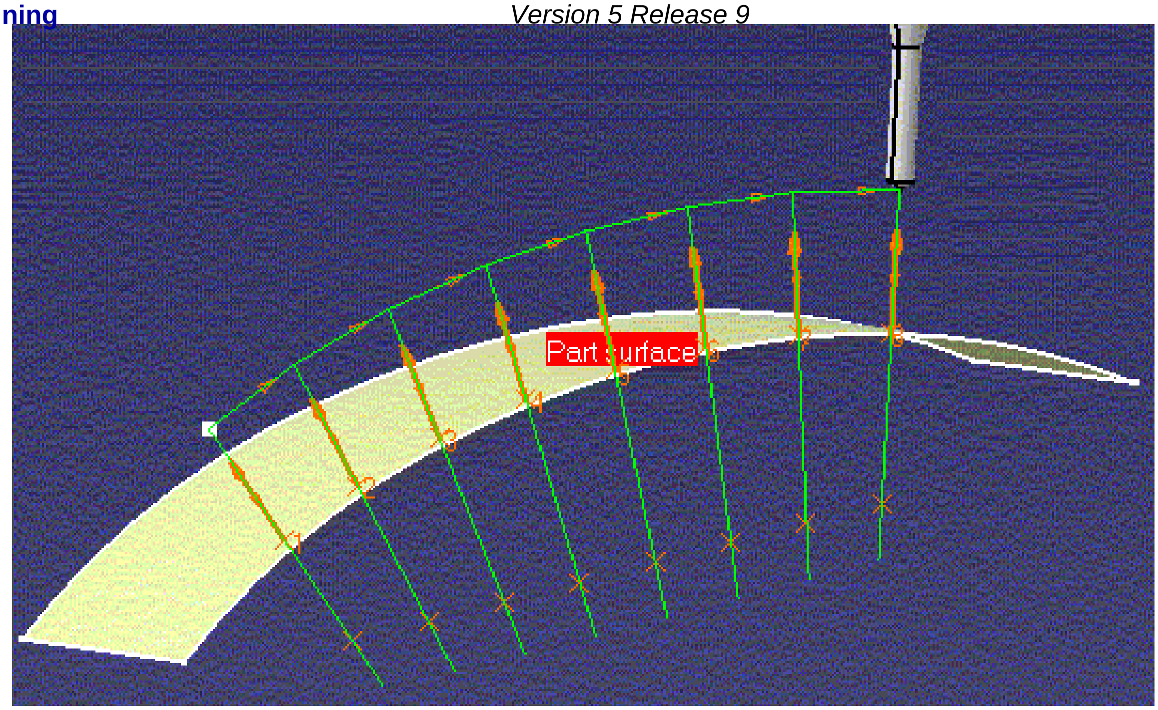
Projection and Top Element Modes

In the Geometry tab page you can choose between Projection and Top Element modes by clicking on the sensitive text.

The following figure illustrates Projection mode. The reference pattern points are projected onto the selected part surface. The projected points and the axes normal to the surface define the hole positions to be drilled.

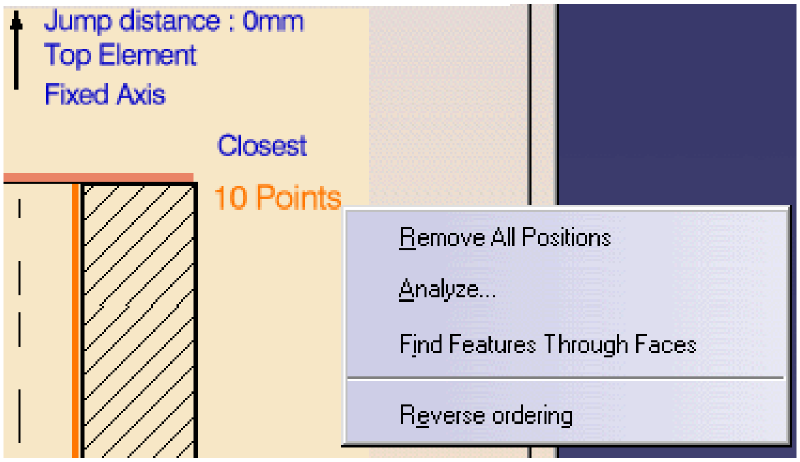


The following figure illustrates Top Element mode. The reference pattern points define the hole positions to be drilled. The machining depth takes into account the normal distance between the reference points and the selected part surface.



Contextual Menu on 'No Points / x Points' Sensitive Text

A number of contextual commands are available for managing hole positions when you right click the 'No Points / x Points' sensitive text in the Geometry tab page.

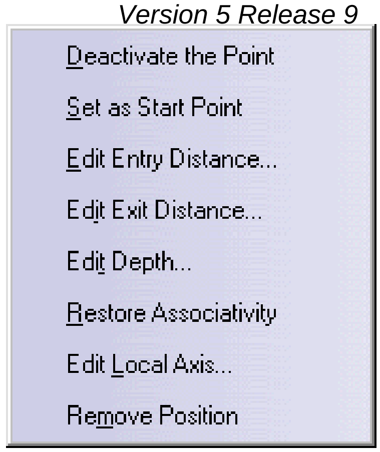


- **Remove All Positions** to reset selected pattern points
- **Analyze** to consult the status of the referenced geometry
- **Find Circular Edges of Faces** to quickly locate circular edges in a selected face
- **Reverse Ordering** to reverse the numbered sequence of pattern points.

Contextual Menu on Pattern Points

A number of contextual commands are available when you right click a pattern point.

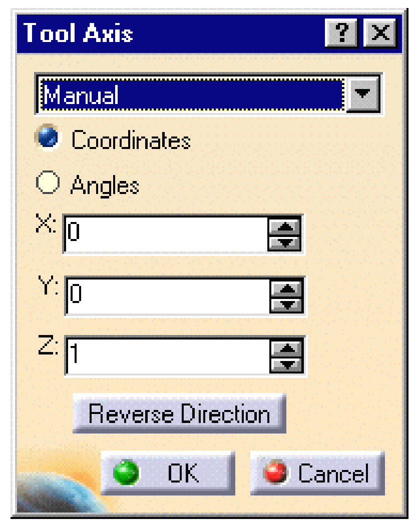
You can locally edit Entry and Exit distances at individual points in a hole pattern. This can be useful for locally specifying a clearance that is greater than the one defined by the approach clearance/jump distance discussed above.



The contextual menu also allows you to:

- deactivate a point in the pattern (a deactivated point can be activated in the pattern again)
- choose the start point for the pattern
- edit the depth of a pattern point
- restore the original values taken from the selected design feature if these were modified by the user, and so restore associativity with the feature
- locally modify the tool axis at a point
- remove a point from the pattern.

Locally Modifying a Tool Axis



Select the Edit Local Axis contextual command, then choose the method for defining tool axis orientation in the Tool Axis dialog box that appears:

- **Manual**. Choose one of the following:
  - **Coordinates** to define the orientation by means of X, Y and Z components
  - **Angles** to define the orientation by means of a rotation of the X, Y or Z axis. The rotation is specified by means of one or two angles.
- **Selection**. If you select a line or linear edge, the tool axis will have the same orientation as that element. If you select a planar element, the tool axis will be normal to that element.
- **Points in the View**. Just select two points to define the orientation.

The tool axis is visualized by means of an arrow. The direction can be reversed by clicking **Reverse Direction** in the dialog box.

Just click OK to accept the specified tool axis orientation.

Origin Offset

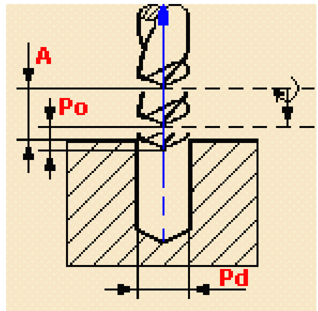
You can specify an **Origin Offset** in order to shift the entire tool path by the specified amount.

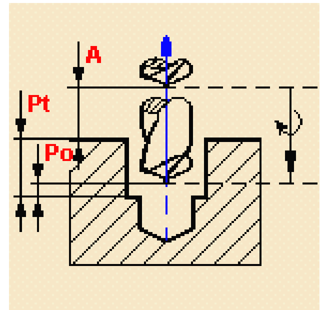
Jump Distance

The jump distance allows an *extra clearance* for moving in Rapid motion between the holes to be drilled whenever this distance is greater than the approach clearance.
For example, for an approach clearance of 2.5mm and a jump distance of 10mm, the extra clearance is 7.5mm.

Plunge Mode

The Plunge mode allows you to specify an axial plunge at plunge feedrate prior to machining.





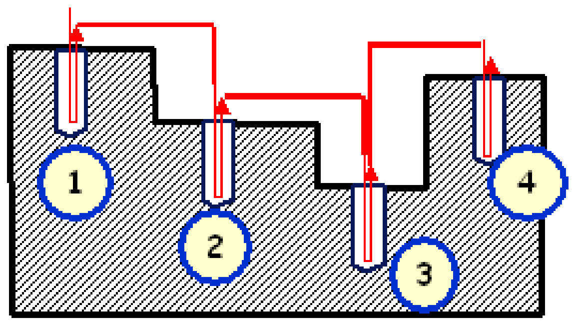
The overall plunge distance is determined as follows:

Approach clearance + (Plunge depth - Plunge offset)

where Plunge depth is determined by a tool tip or tool diameter value.

Holes at Different Levels

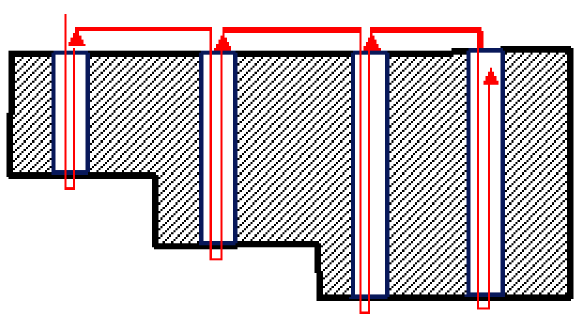
For 2.5-axis operations, the program automatically manages holes at different levels using horizontal transition paths.



Machining Different Depths

When dealing with design feature holes in design patterns, both the result and specification mode are taken into account (except spot drilling, counterboring, countersinking and threading operations).

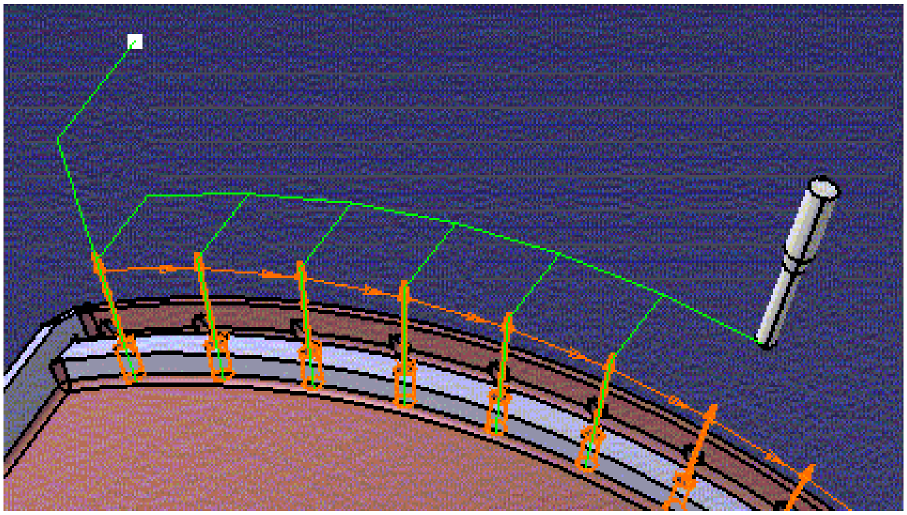
Select the **Machine different depths** checkbox when you want the program to automatically manage different depths of holes in a pattern (result mode).



If the checkbox is not selected, the program uses the values specified in the Geometry tab page for the pattern holes (specification mode).

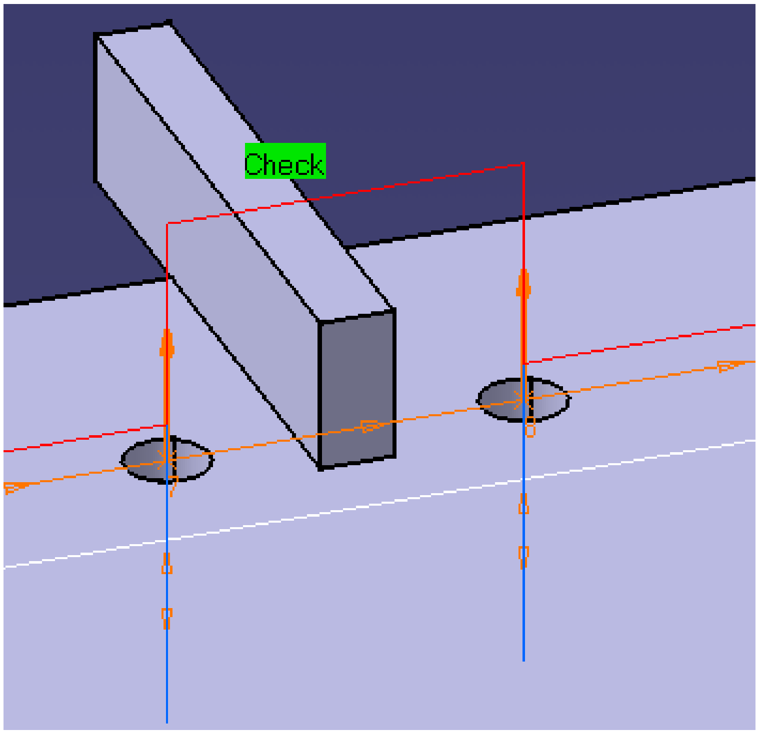
Macros in Drilling Operations

The Macro Tab page in the operation definition dialog box allows customized transitions paths for approach, retract and linking. Refer to Define Macros on an Axial Machining Operation for more information.



Collision Check on Macros used in Drilling Operations

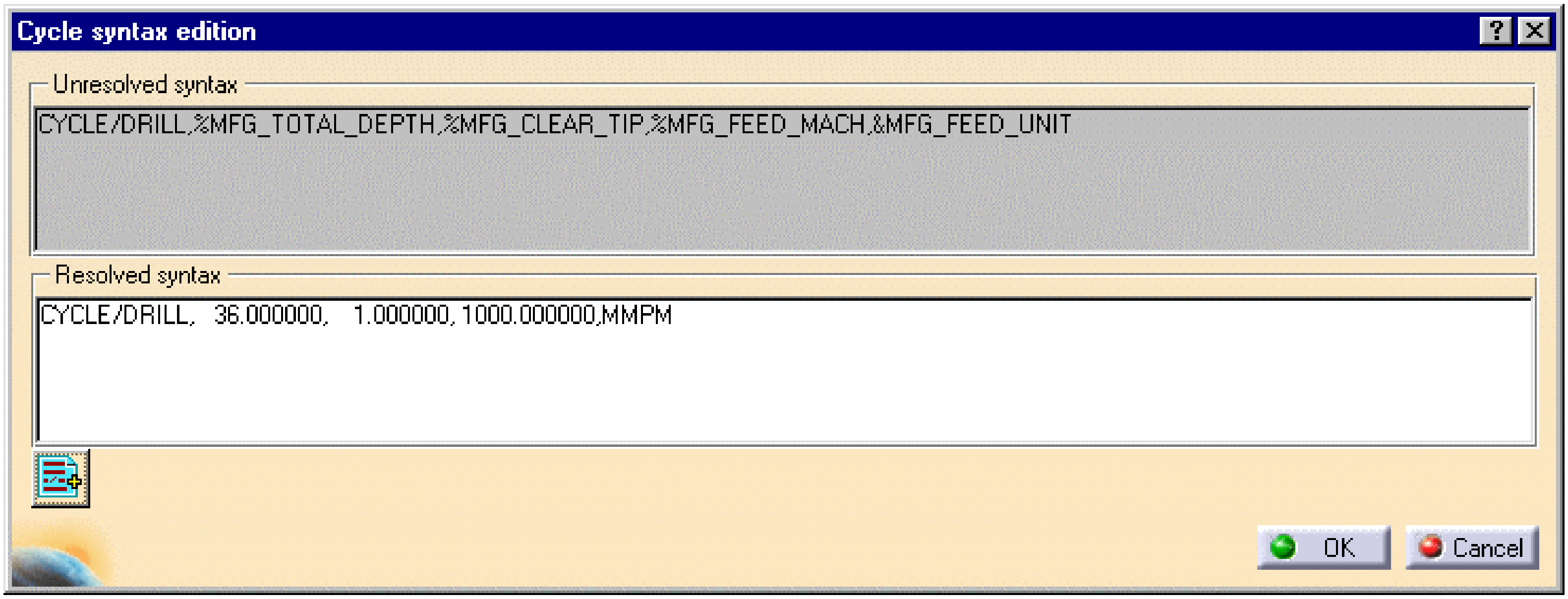
All types of macros used in Drilling Operations are collision checked. If a check element is specified between two machined positions, a linking macro is applied to avoid collisions. Check (or fixture) elements as well as an associated **Offset on Check** can be specified in the Geometry tab page.



Editing CYCLE Syntaxes

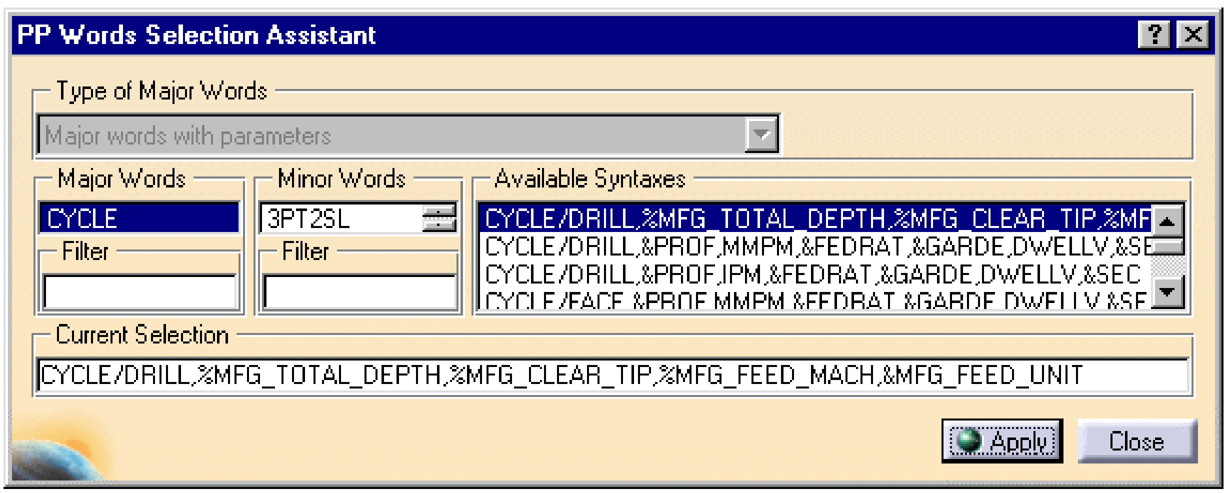
For all axial operations the **Edit Cycle** command in the Axial Machining Operation dialog box allows you to:

- display the unresolved syntax of the NC Instruction of the operation. This is the syntax as specified in the PP table referenced by the current part Operation.
- display and, if needed, modify the syntax that is resolved either by geometric selection and user entries.



You can access all the CYCLE syntaxes contained in the current PP table by means of the PP instruction icon.

You can then select the desired syntax to be used by means of the procedure described in the Insert PP Instruction section.



Output Syntax

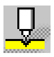
If you want to generate CYCLE statements, you must select the **Output CYCLE syntax** checkbox in the Strategy tab page and set the Syntax Used option to Yes in the NC Output generation dialog box.
Otherwise, GOTO statements will be generated.

Note that when several axis orientations are present in a machining pattern, output of the components of the tool axis orientation is possible whenever the NC data format is set to Axis (X, Y, Z, I, J, K) in the Part Operation's Machine Editor dialog box.

Supported Axial Machining Operation Types

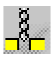
The following axial machining operations can be included in your manufacturing program.

Spot Drilling Operation

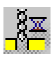


Create a Spot Drilling Operation: Select the Spot Drilling icon then select the positions to be spot drilled and specify the tool to be used. Specify the depth mode (By Tip or By Diameter). Finally, set the other machining parameters, macros, and feeds and speeds as needed.

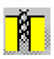
Drilling Operations



Create a Drilling Operation: Select the Drilling icon then select the hole or hole pattern to be drilled and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder). Finally set the other machining parameters, macros, and feeds and speeds as needed.



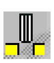
Create a Drilling Dwell Delay Operation: Select the Drilling Dwell Delay icon then select the hole or hole pattern to be drilled and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder) and the dwell duration. Finally set the other machining parameters, macros, and feeds and speeds as needed.



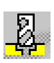
Create a Drilling Deep Hole Operation: Select the Drilling Deep Hole icon then select the hole or hole pattern to be drilled and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder). Finally set the other machining parameters, macros, and feeds and speeds as needed.

Create a Drilling Break Chips Operation: Select the Drilling Break Chips icon then select the hole or hole pattern to be drilled and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder). Finally set the other machining parameters, macros, and feeds and speeds as needed.

Hole Finishing Operations

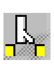


Create a Reaming Operation: Select the Reaming icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder). Finally set the other machining parameters, macros, and feeds and speeds as needed.

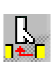


Create a Counterboring Operation: Select the Counterboring icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder). Finally set the other machining parameters, macros, and feeds and speeds as needed.

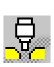
Boring Operations



Create a Boring Operation: Select the Boring icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder). Finally set the other machining parameters, macros, and feeds and speeds as needed.



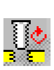
Create a Boring Spindle Stop Operation: Select the Boring Spindle Stop icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder) and the amount of tool shift away from the bored hole. Finally set the other machining parameters, macros, and feeds and speeds as needed.



Create a Boring and Chamfering Operation: Select the Boring and Chamfering icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder). Finally, for the boring and chamfering phases set the machining parameters, macros, and feeds and speeds as needed.

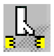
Create a Back Boring Operation: Select the Back Boring icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder) and the amount of tool shift for back boring. Finally set the other machining parameters, macros, and feeds and speeds as needed.

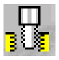
Threading Operations



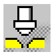
Create a Tapping Operation: Select the Tapping icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder). Finally set the other machining parameters, macros, and feeds and speeds as needed.

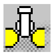

Create a Reverse Threading Operation: Select the Reverse Threading icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify the depth mode (By Tip or By Shoulder). Finally set the other machining parameters, macros, and feeds and speeds as needed.


Create a Thread without Tap Head Operation: Select the Thread without Tap Head icon then select the hole or hole pattern to be machined and specify the tool to be used. The By Tip depth mode is imposed for this operation. Finally set the other machining parameters, macros, and feeds and speeds as needed.

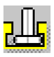

Create a Thread Milling Operation: Select the Thread Milling icon then select the hole or hole pattern to be machined and specify the tool to be used. Finally set the other machining parameters, macros, and feeds and speeds as needed.

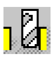
Countersinking and Chamfering Operations


Create a Countersinking Operation: Select the Countersinking icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify the depth mode (By Distance or By Diameter). Finally set the other machining parameters, macros, and feeds and speeds as needed.


Create a Chamfering Two Sides Operation: Select the Chamfering Two Sides icon then select the hole or hole pattern to be machined and specify the tool to be used. The By Tip depth mode is imposed for this operation. Finally set the other machining parameters, macros, and feeds and speeds as needed.

T-Slotting and Circular Milling


Create a T-Slotting Operation: Select the T-Slotting icon then select the hole or hole pattern to be machined and specify the tool to be used. The By Tip depth mode is imposed for this operation. Finally set the other machining parameters, macros, and feeds and speeds as needed.


Create a Circular Milling Operation: Select the Circular Milling icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify the machining mode (Standard or Helical). Finally set the other machining parameters, macros, and feeds and speeds as needed.

Create a Spot Drilling Operation

This task shows how to insert a Spot Drilling operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

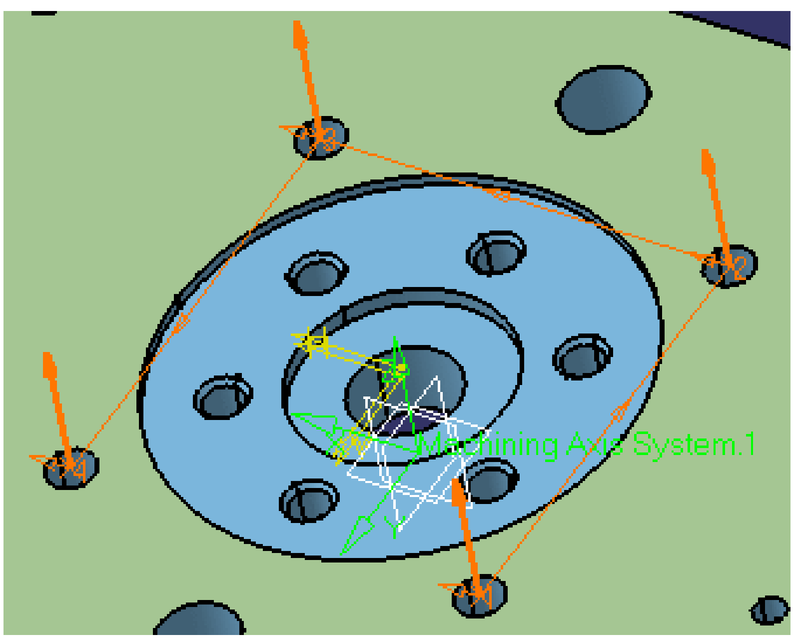
Make the Manufacturing Program current in the specification tree.

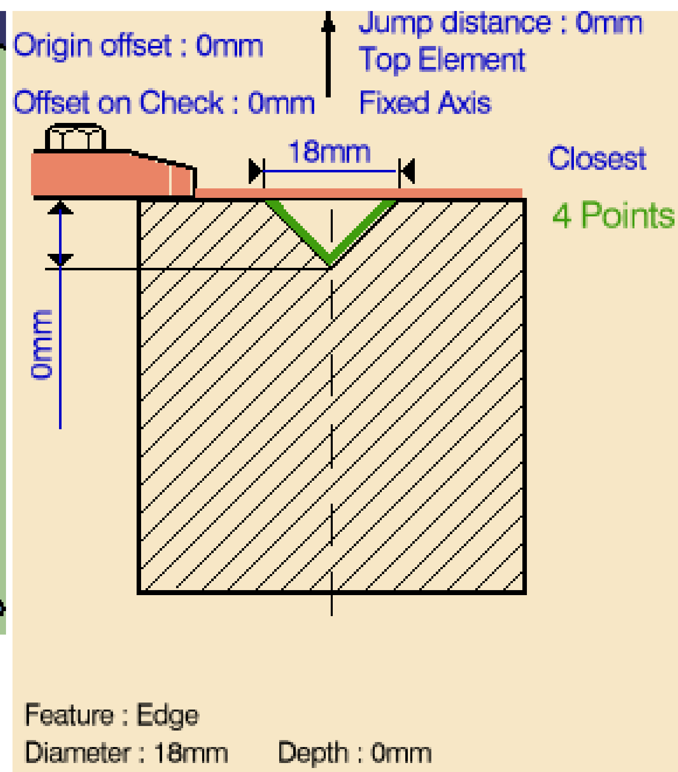
1. Select the Spot Drilling icon .

A Spot Drilling entity along with a default tool is added to the program.

The Spot Drilling dialog box appears directly at the Geometry tab page . This tab page includes an icon representing a simple hole. There are several hot spots in the icon.

2. Select red hole depth representation, then select the points to be spot drilled.
You can do this by selecting the circular edges of holes.
In this case, the circle centers are taken as the points to be spot drilled.
Just double click to end your selections.

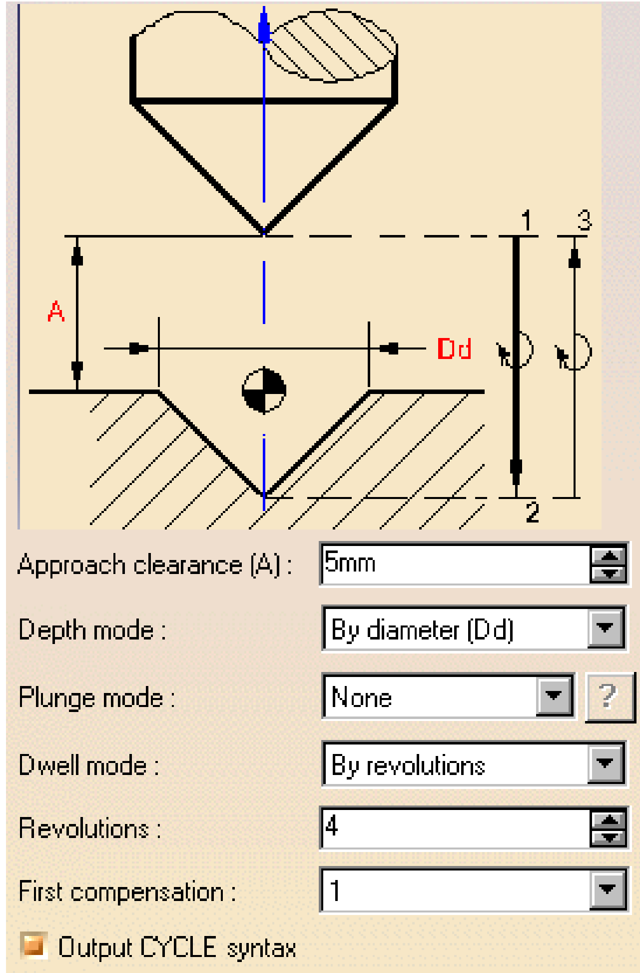




3. If needed, click on the tool axis symbol to select a tool axis direction.

4. Select the Strategy tab page to specify the following machining parameters:
   - approach clearance
   - depth mode: by diameter
     The diameter value used is the one specified in the geometry tab page.
   - dwell
   - compensation number depending on those available on the tool.

   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion is as follows:
   - Motion at machining feedrate from 1 to 2
   - Dwell for the specified duration
   - Retract at retract feedrate from 2 to 3.
6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Spot Drilling operations:

```
CYCLE / SPDRL, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE, &MFG_FEED_UNIT,
        %MFG_CLEAR_TIP, DWELL, %MFG_DWELL_REVOL
```

A typical NC data output is as follows:

```
CYCLE/SPDRL, 25.000000, 500.000000, MMPM, 5.000000, DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_SPOT_DRILLING section of the Manufacturing Infrastructure User's Guide.

Create a Drilling Operation

This task shows how to insert a Drilling operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths) .

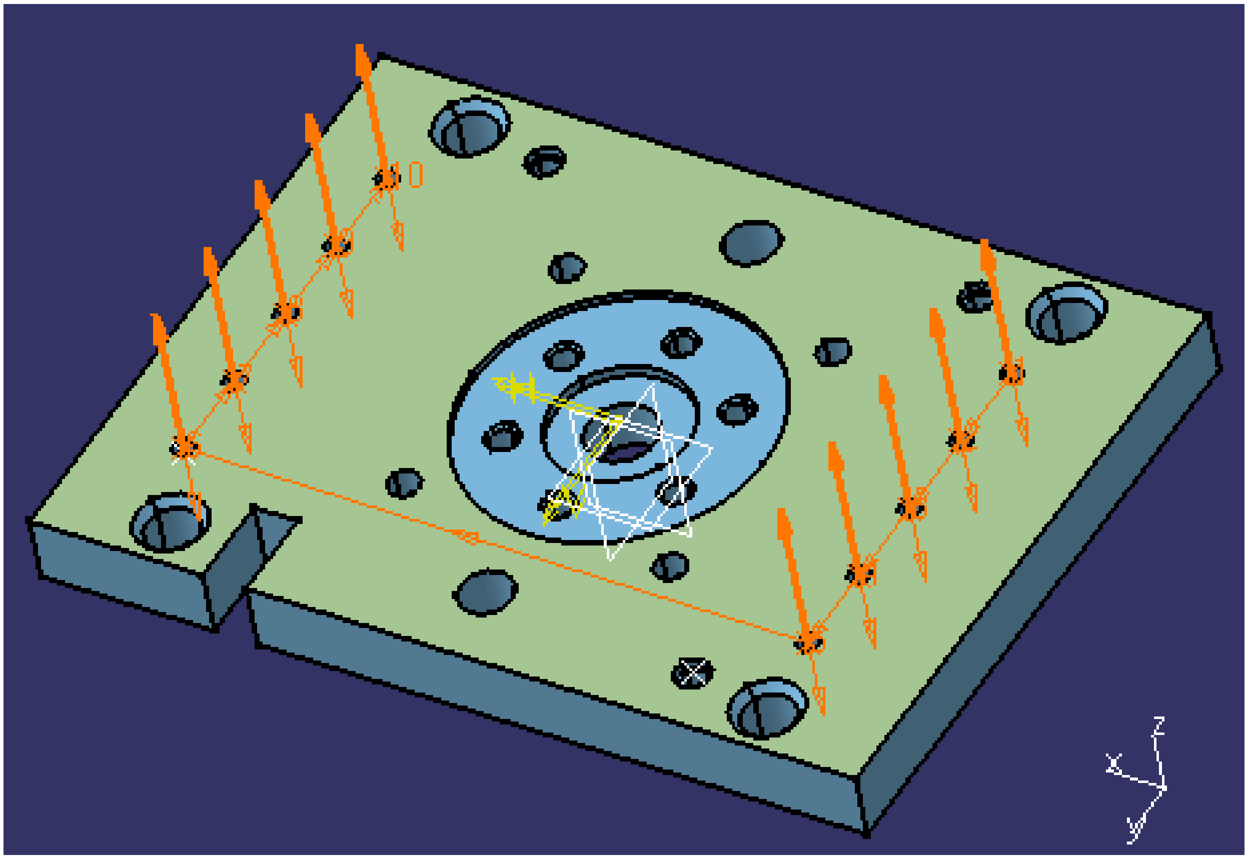
Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the Drilling icon .

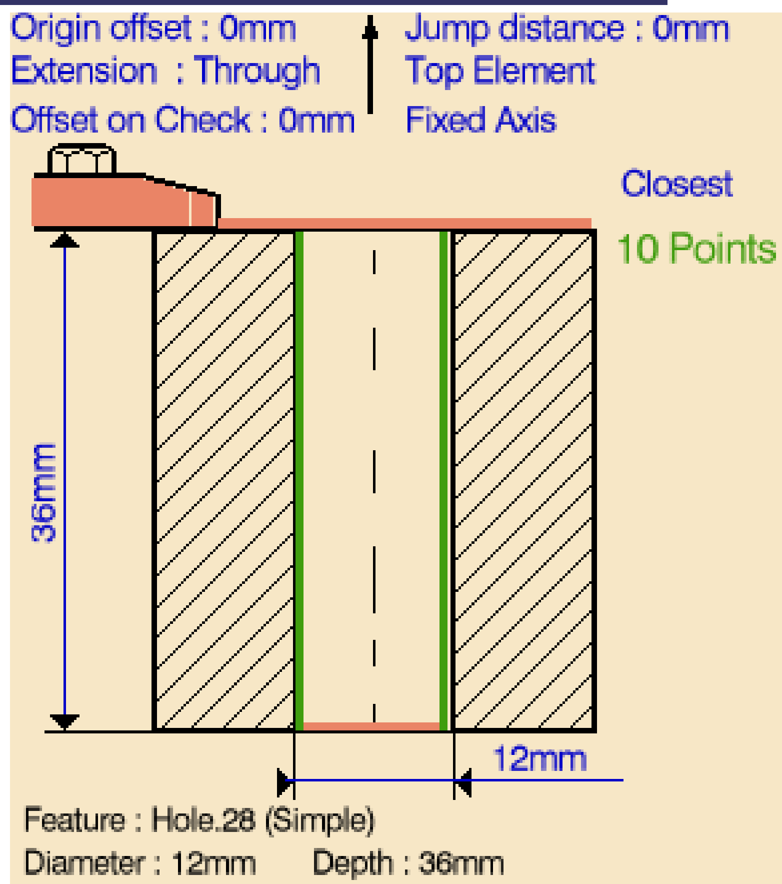
   A Drilling entity along with a default tool is added to the program.

   The Drilling dialog box appears directly at the Geometry tab page . This tab page includes a sensitive icon to help you specify the geometry of the hole or hole pattern to be machined.
2. Select the red hole depth representation then select the pattern of 10 holes as shown below. Just double click to end your selections.

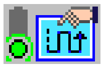


The sensitive icon is updated with the following information:

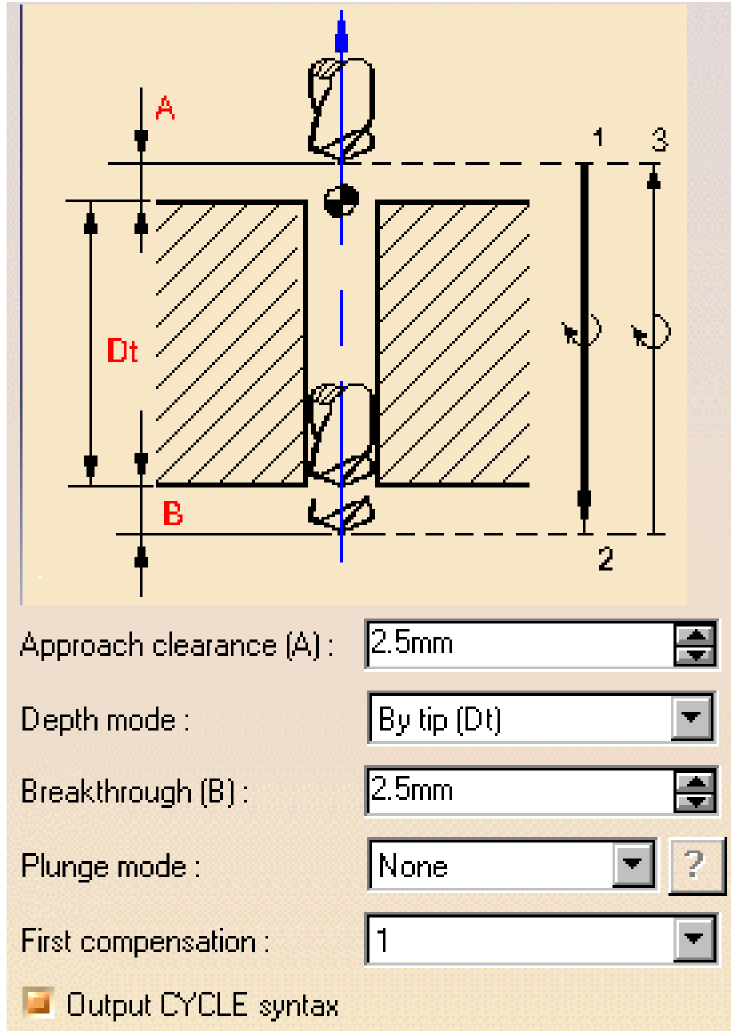
- depth and diameter of the first selected feature
- hole extension type: through hole
- number of points to machine.



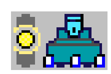
3. If needed, you can define the tool axis direction by first selecting the axis representation in the sensitive icon then specifying the direction by means of the dialog box that appears.

4. If needed, you can define a clearance by first double clicking the Jump Distance parameter in the sensitive icon then specifying a value in the Edit Parameter dialog box that appears.
5. Select the Strategy tab page  to specify the following machining parameters:
   - Approach clearance
   - Depth mode: by tip
     The depth value used is the one specified in the Geometry tab page.
   - Breakthrough distance
   - Compensation number depending on those available on the tool.

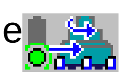
   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use. This is described in Edit the Tool of an Operation.

Remember that you can make use of the hole diameter found on the selected hole feature to select an appropriate tool.

6. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

   Note that in the Drilling tool path represented in the strategy page, tool motion is as follows:
   - machining feedrate from 1 to 2
   - retract or rapid feedrate from 2 to 3.

7. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

Before accepting the operation, you should check its validity by replaying the tool path.

8. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Drilling operations:

```
CYCLE/DRILL, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT,
             %MFG_CLEAR_TIP
```

A typical NC data output is as follows:

```
CYCLE/DRILL, 38.500000, 500.000000, MMPM, 2.500000
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_DRILLING section of the Manufacturing Infrastructure User's Guide.

Create a Drilling Dwell Delay Operation

This task shows how to insert a Drilling Dwell Delay operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

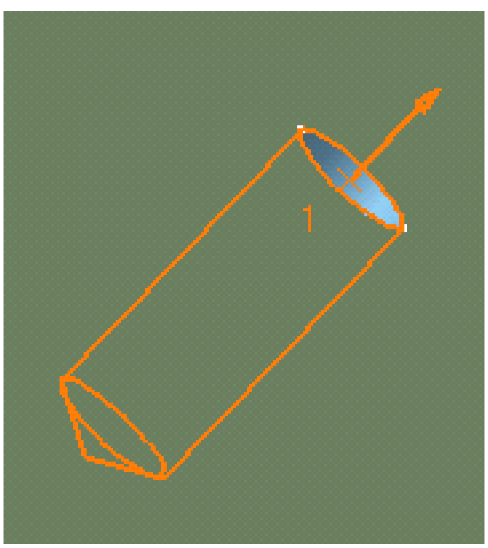
Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the Drilling Dwell Delay icon .

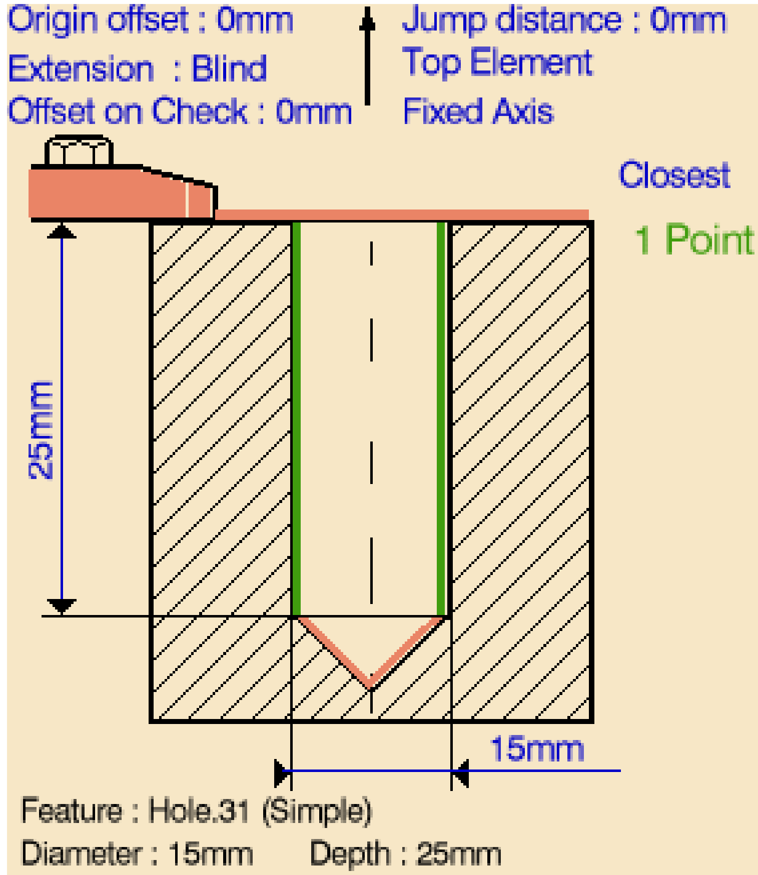
   A Drilling Dwell Delay entity along with a default tool is added to the program.

   The Drilling Dwell Delay dialog box appears directly at the Geometry tab page . This tab page includes a sensitive icon to help you specify the geometry of the hole or hole pattern to drill.
2. Select the red hole depth representation then select the hole feature as shown. Just double click to end your selection.



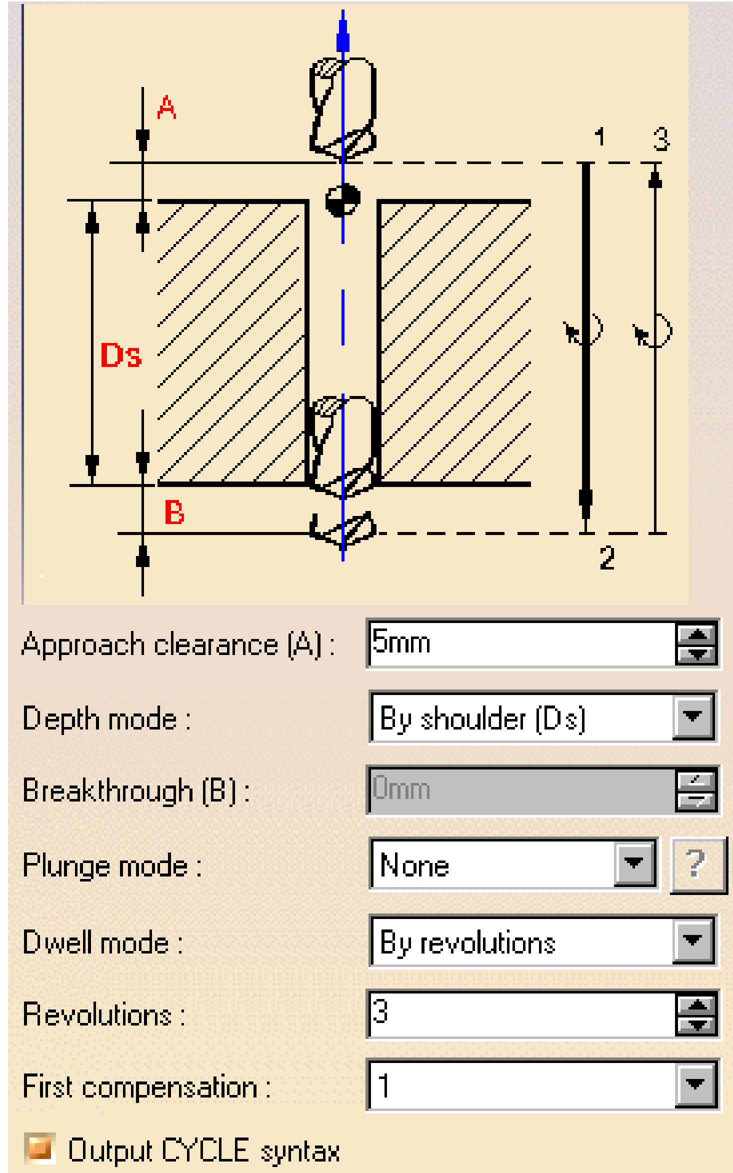
The sensitive icon is updated with the following information:

- depth and diameter of the selected hole
- hole extension type: blind.



3. If needed, you can define the tool axis direction by first selecting the axis representation in the sensitive icon then specifying the direction by means of the dialog box that appears.
4. Select the Strategy tab page to specify the following machining strategy parameters:
   - Approach clearance
   - Depth mode: by shoulder
     The depth value used is the one specified in the Geometry tab page.
   - Dwell delay
   - Compensation number depending on those available on the tool.

   The other parameters are optional in this case.

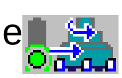


A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

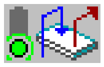
Remember that you can make use of the hole diameter found on the selected hole feature to select an appropriate tool.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion is as follows:

   - machining feedrate from 1 to 2
   - dwell for the specified duration
   - retract or rapid feedrate from 2 to 3.

6 If you want to specify approach and retract motion for the operation, select the Macros tab page  to specify the desired transition paths.

The general procedure for this is described in Define Macros of an Axial Machining Operation.

Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Drilling Dwell Delay operations:

```
CYCLE / DRILL, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT,
               %MFG_CLEAR_TIP, DWELL, %MFG_DWELL_REVOL
```

A typical NC data output is as follows:

```
CYCLE/DRILL, 25.000000, 500.000000, MMPM, 5.000000, DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_DRILLING_DWELL_DELAY section of the Manufacturing Infrastructure User's Guide.

Create a Drilling Deep Hole Operation

This task shows how to insert a Drilling Deep Hole operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

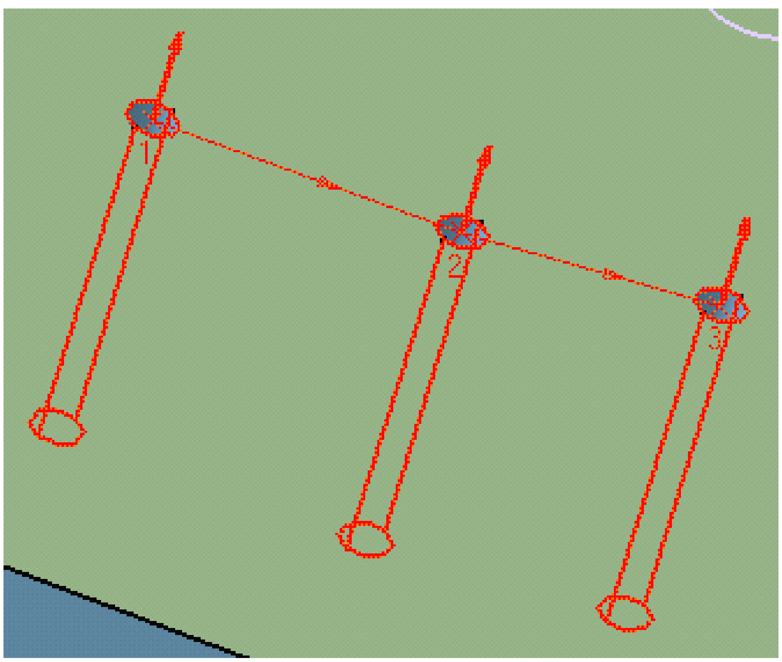
Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

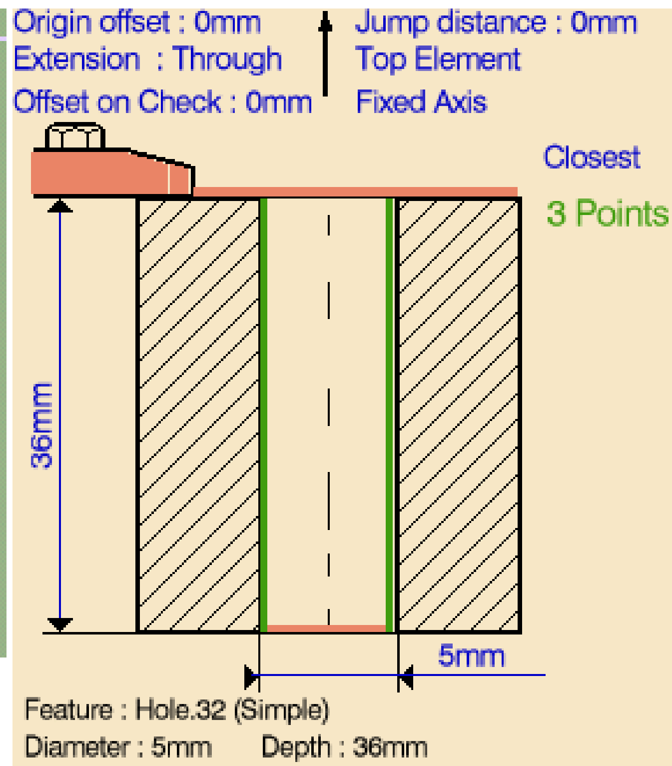
Make the Manufacturing Program current in the specification tree.

1. Select the Drilling Deep Hole icon.

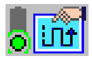
   A Drilling Deep Hole entity along with a default tool is added to the program.

   The Drilling Deep Hole dialog box appears directly at the Geometry tab page. This tab page includes a sensitive icon to help you specify the geometry of the hole or hole pattern to be machined.
2. Select the red hole depth representation then select the hole features as shown below.
   Just double click to end your selections.

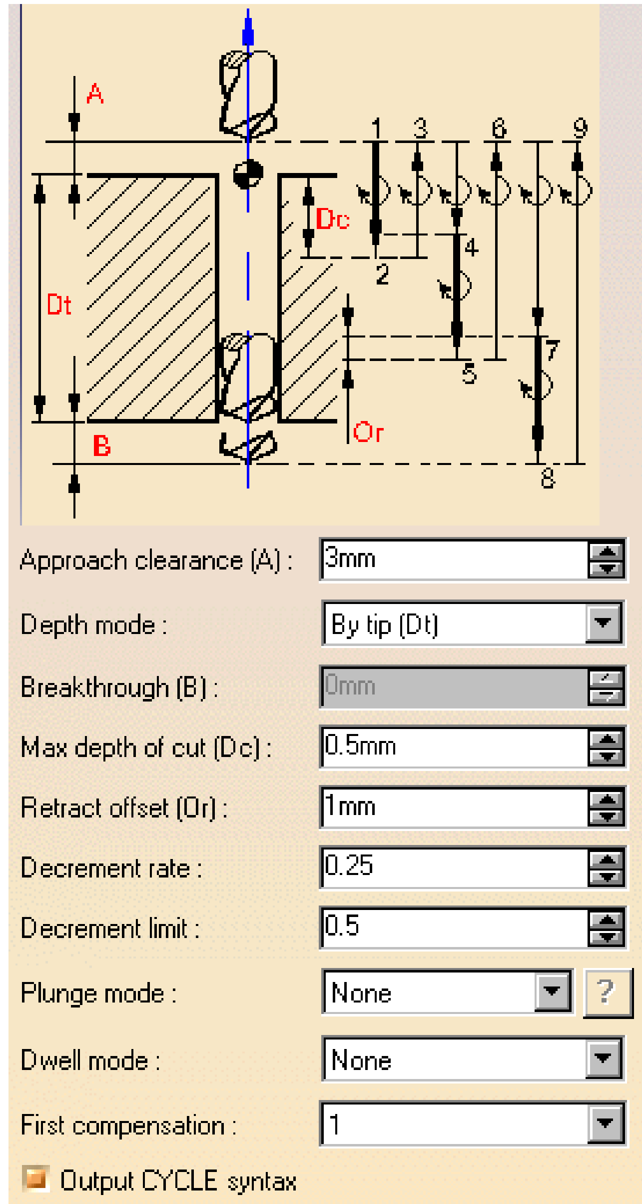




   The sensitive icon is updated with the following information:
   - depth and diameter of the first selected hole
   - hole extension type: through
   - number of points to machine.
3. If needed, you can define the tool axis direction by first selecting the axis representation in the sensitive icon then specifying the direction by means of the dialog box that appears.

4. Select the Strategy tab page  to specify the following machining parameters:
   - Approach clearance
   - Depth mode: by tip
     The depth value used is the one specified in the Geometry tab page.
   - Breakthrough distance
   - Maximum depth of cut and retract offset
   - Decrement rate and limit
   - Dwell
   - Compensation number depending on those available on the tool.

   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

Remember that you can make use of the hole diameter found on the selected hole feature to select an appropriate tool.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion is as follows:

   - Motion at machining feedrate from 1 to 2
   - Dwell for specified duration
   - Retract at retract feedrate from 2 to 3
   - Motion at rapid rate from 3 to 4
   - Motion at machining feedrate from 4 to 5
   - Dwell for specified duration
   - Retract at retract feedrate from 5 to 6
   - Motion at rapid rate from 6 to 7
   - Motion at machining feedrate from 7 to 8
   - Dwell for specified duration
   - Retract at retract feedrate from 8 to 9

   Distance (1,2) = A + Dc
   Distance (3,4) = A + Dc - Or
   Distance (4,5) = Or + Dc*(1 - decrement rate)
   Distance (7,8) = Or + Dc*(1 - 2*decrement rate).

6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Drilling Deep Hole operations:

```
CYCLE/DEEPHL,%MFG_TOTAL_DEPTH,INCR,%MFG_AXIAL_DEPTH,%MFG_FEED_MACH_VALUE,
             &MFG_FEED_UNIT,%MFG_CLEAR_TIP,DWELL,%MFG_DWELL_REVOL
```

A typical NC data output is as follows:

```
CYCLE/DEEPHL, 25.000000, INCR, 5.000000, 500.000000, MMPM, 5.000000,
DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_DEEPHOLE section of the Manufacturing Infrastructure User's Guide.

Create a Drilling Break Chips Operation

This task shows how to insert a Drilling Break Chips operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

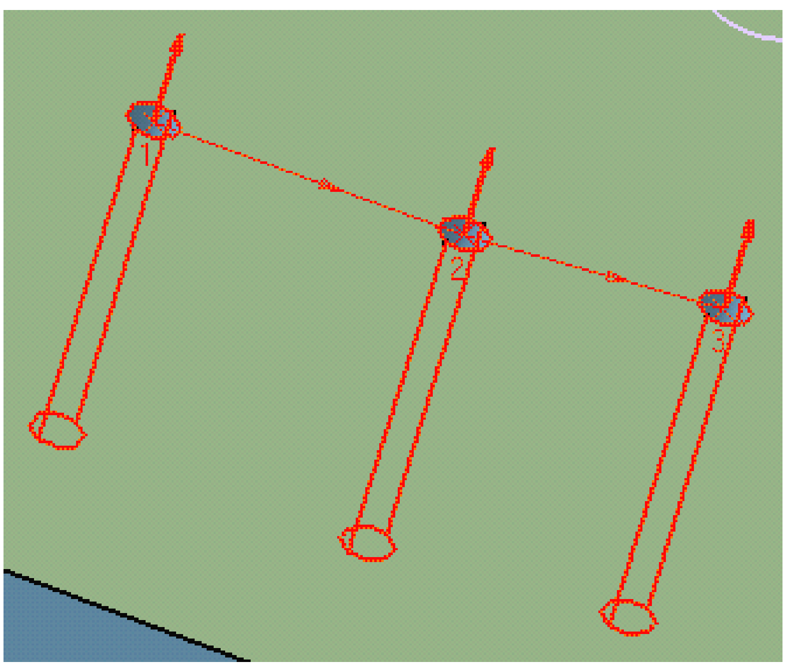
Make the Manufacturing Program current in the specification tree.

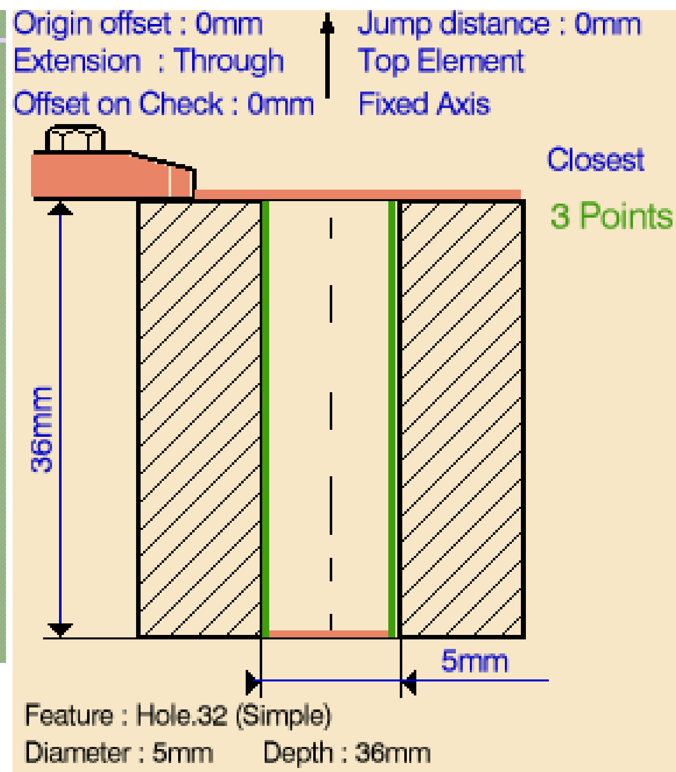
1. Select the Drilling Break Chips icon.

A Drilling Break Chips entity along with a default tool is added to the program.

The Drilling Break Chips dialog box appears directly at the Geometry tab page. This tab page includes a sensitive icon to help you specify the geometry of the hole or hole pattern to be machined.

2. Select the red hole depth representation then select the hole feature as shown below.
Just double click to end your selections.





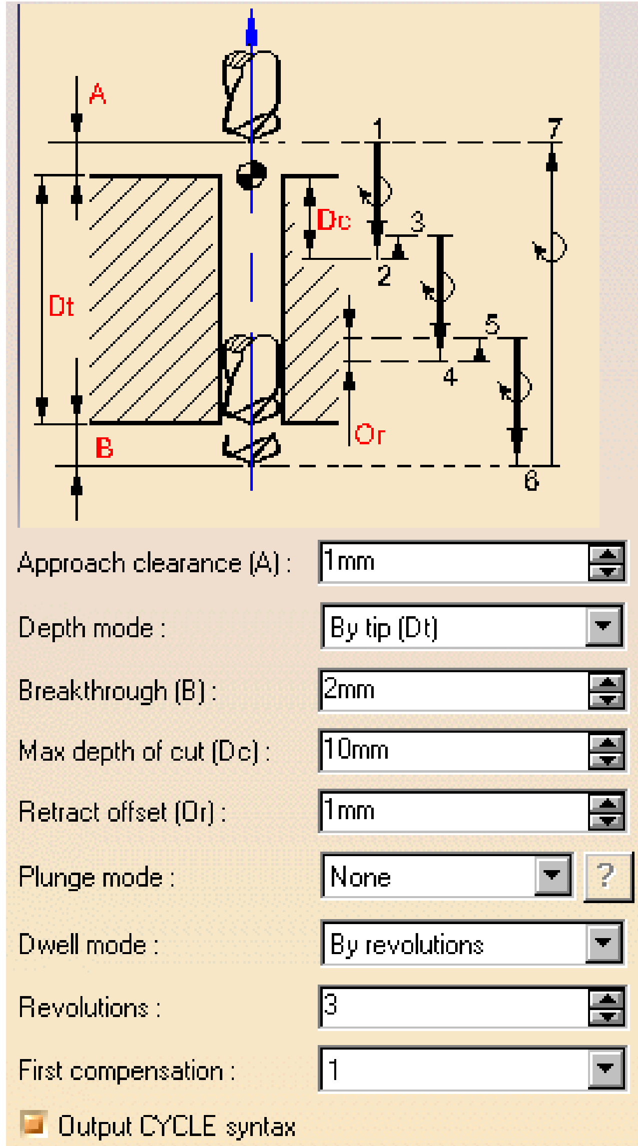
The sensitive icon is updated with the following information:

- depth and diameter of the selected hole
- hole extension type: through.

3. If needed, you can define the tool axis direction by first selecting the axis representation in the sensitive icon then specifying the direction by means of the dialog box that appears.

4. Select the Strategy tab page to specify the following machining parameters.
   - Approach clearance
   - Depth mode: by tip
     The depth value used is the one specified in the geometry tab page.
   - Breakthrough distance
   - Maximum depth of cut and retract offset
   - Dwell
   - Compensation number depending on those available on the tool.

   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

Remember that you can make use of the hole diameter found on the selected hole feature to select an appropriate tool.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion is as follows:
   - Motion at machining feedrate from 1 to 2
   - Dwell for specified duration
   - Retract at retract feedrate from 2 to 3
   - Motion at machining feedrate from 3 to 4
   - Dwell for specified duration
   - Retract at retract feedrate from 4 to 5
   - Motion at machining feedrate from 5 to 6
   - Dwell for specified duration

- Retract at retract feedrate from 6 to 7

Distance (1,2) = A + Dc
Distance (2,3) = Distance (4,5) = Or
Distance (3,4) = Distance (5,6) = Or + Dc

6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Drilling Break Chips operations:

```
CYCLE/BRKCHP,%MFG_TOTAL_DEPTH,INCR,%MFG_AXIAL_DEPTH,%MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT,
             %MFG_CLEAR_TIP,DWELL,%MFG_DWELL_REVOL
```

A typical NC data output is as follows:

```
CYCLE/BRKCHP, 25.000000, INCR, 5.000000, 500.000000, MMPM, 5.000000,
DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_BREAK_CHIPS section of the Manufacturing Infrastructure User's Guide.

Create a Reaming Operation

This task shows how to insert a Reaming operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

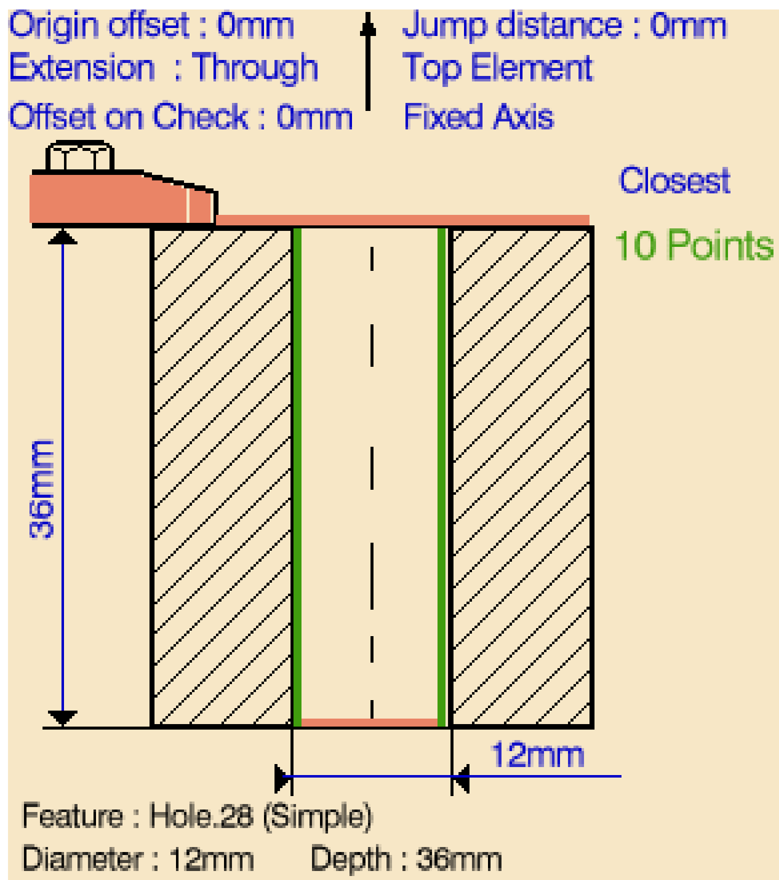
Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the Reaming icon .

   A Reaming entity along with a default tool is added to the program.

   The Reaming dialog box appears directly at the Geometry tab page . This tab page includes a sensitive icon to help you specify the geometry of the hole or hole pattern to be machined.



2. Select the red hole depth representation then select the pattern of 10 holes.
   Just double click to end your selections.

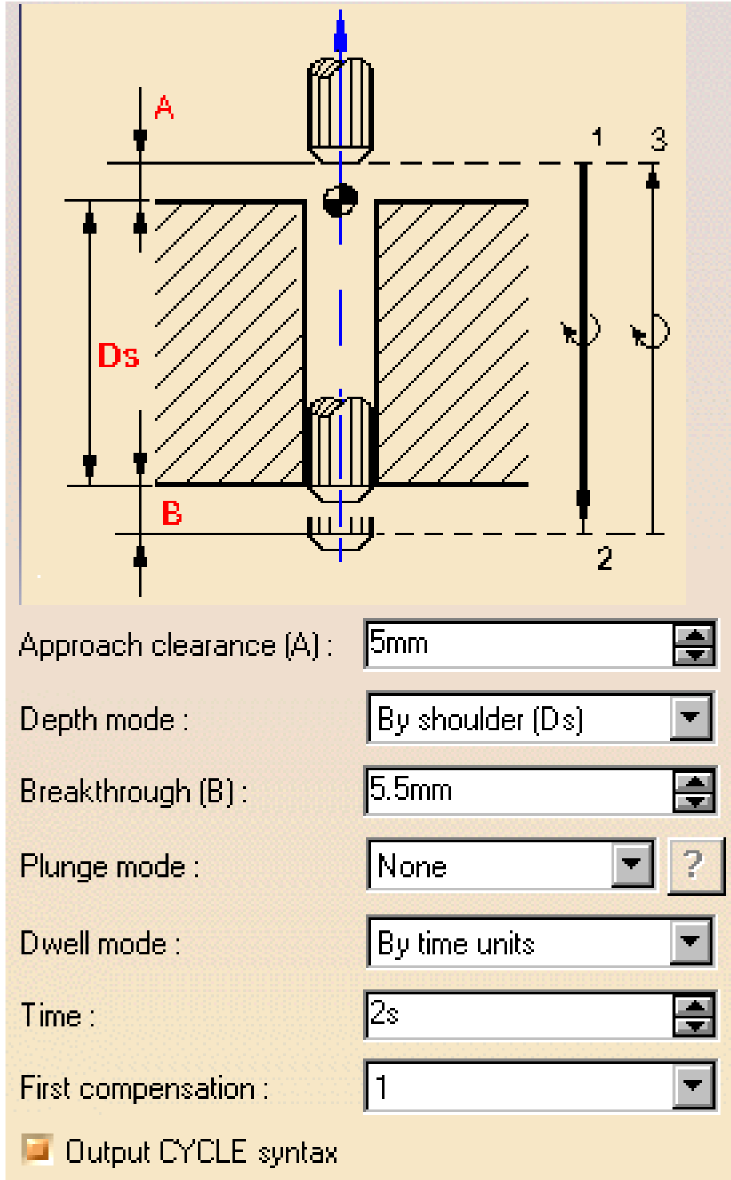
   The sensitive icon is updated with the following information:

   - depth and diameter of the first selected feature
   - hole extension type: through hole
   - number of points to machine.

3. If needed, select the tool axis direction.

4. Select the Strategy tab page to specify the following machining parameters.
   - Approach clearance
   - Depth mode: by shoulder
     The depth value used is the one specified in the Geometry tab page.
   - Dwell (in seconds)
   - Compensation number depending on those available on the tool.

   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion is at:
   - Motion at machining feedrate from 1 to 2
   - Dwell for specified duration
   - Retract at retract feedrate from 2 to 3.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Reaming operations:

```
CYCLE/REAM, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT,
            %MFG_CLEAR_TIP, DWELL, %MFG_DWELL_REVOL
```

A typical NC data output is as follows:

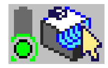
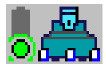
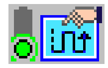
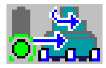
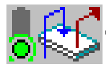
```
CYCLE/REAM, 25.000000, 500.000000, MMPM, 5.000000, DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_REAMING section of the Manufacturing Infrastructure User's Guide.

Create a Counterboring Operation

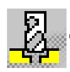
This task shows how to insert a Counterboring operation in the program.

To create the operation you must define:

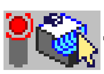
- the geometry of the holes to be machined 
- the tool that will be used 
- the parameters of the machining strategy 
- the feedrates and spindle speeds 
- the macros (transition paths) .

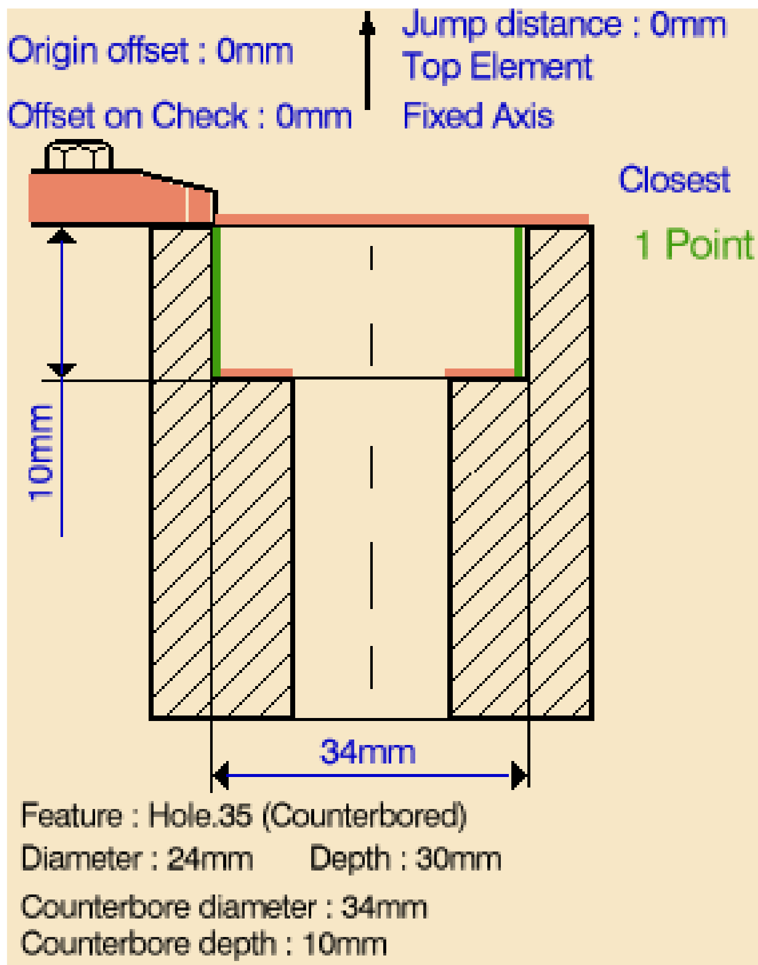
Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the Counterboring icon .

   A Counterboring entity along with a default tool is added to the program.

   The Counterboring dialog box appears directly at the Geometry tab page .

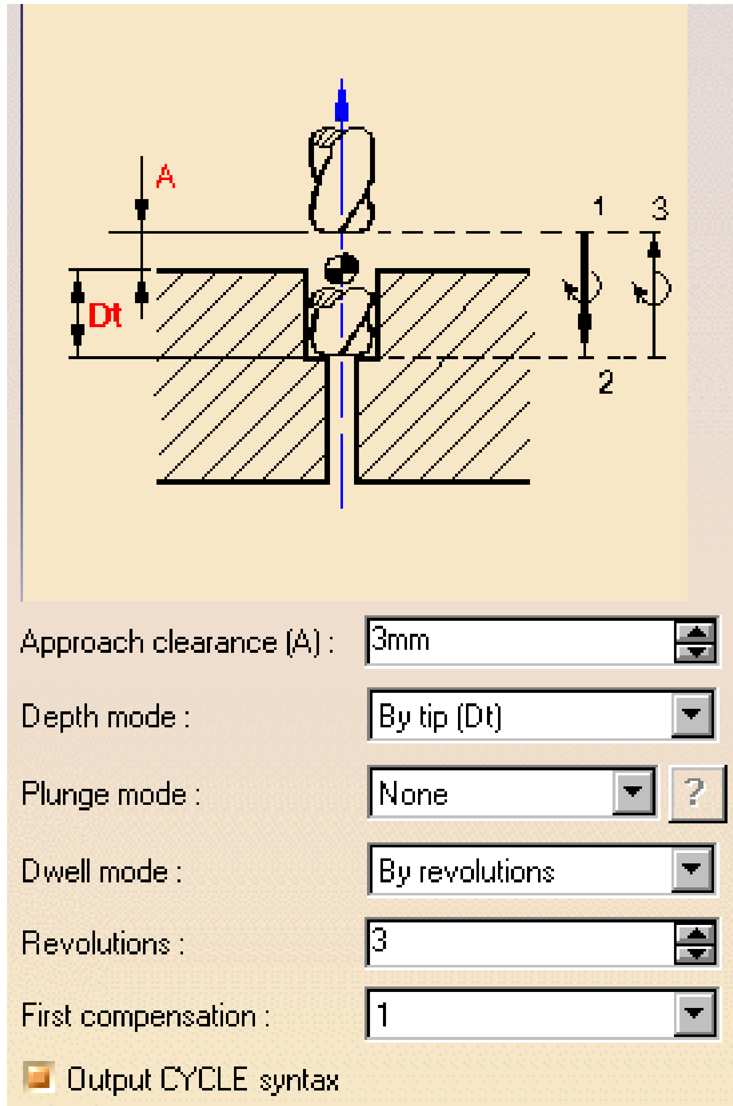


2. Select the red hole depth representation then select hole geometry in the 3D window.
   Just double click to end your selection.

3. If needed, select the tool axis direction.

4. Select the Strategy tab page and specify the following machining parameters.
   - Approach clearance
   - Depth mode: by tip
     The depth value used is the one specified in the Geometry tab page.
   - Dwell
   - Compensation number depending on those available on the tool.

   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the toolpath represented in the strategy page, tool motion is at:
   - Motion at machining feedrate from 1 to 2
   - Dwell for specified duration
   - Retract at retract feedrate from 2 to 3.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Counterboring operations:

```
CYCLE/CBORE, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE, 
&MFG_FEED_UNIT,
             %MFG_CLEAR_TIP, DWELL, %MFG_DWELL_REVOL
```

A typical NC data output is as follows:

```
CYCLE/CBORE, 25.000000, 500.000000, MMPM, 5.000000, DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_COUNTERBORING section of the Manufacturing Infrastructure User's Guide.

Create a Boring Operation

This task shows how to insert a Boring operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the Boring icon.

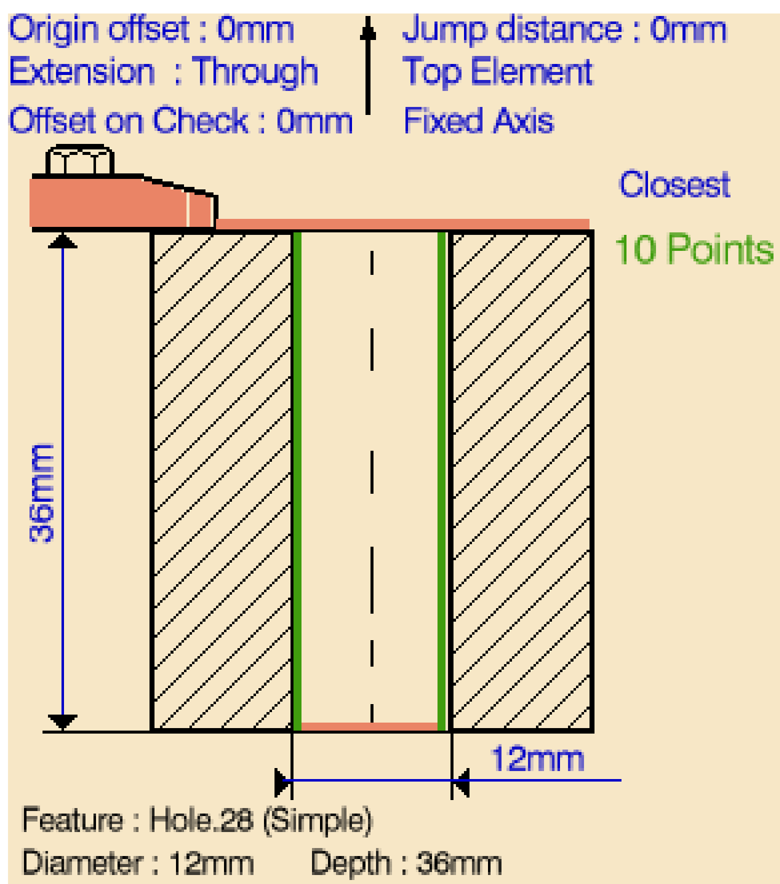
   A Boring entity along with a default tool is added to the program.

   The Boring dialog box appears directly at the Geometry tab page.

2. Select the red hole depth representation then select the pattern of 10 holes.
   Just double click to end your selections.

   The sensitive icon is updated with the following information:

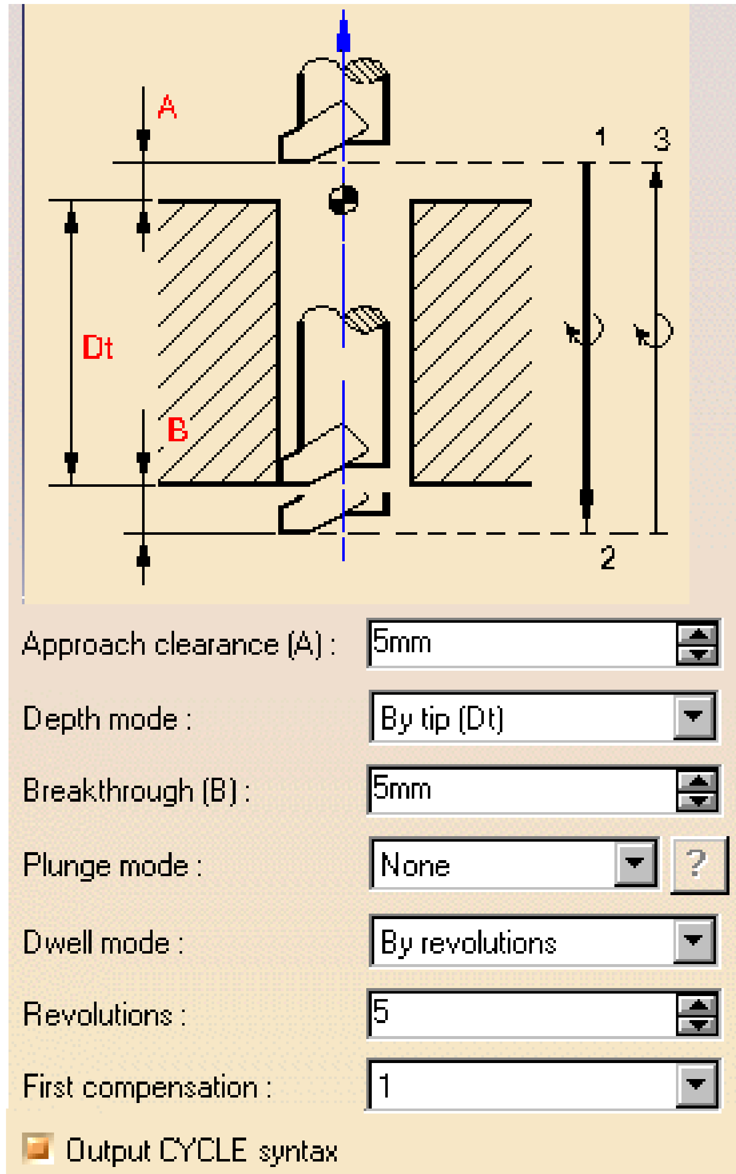
   - depth and diameter of the first selected feature
   - hole extension type: through hole
   - number of points to machine.



3. If needed, select the tool axis direction.

4. Select the Strategy tab page to specify the following machining parameters:
   - approach clearance
   - depth mode: by tip
     The depth value used is the one specified in the Geometry tab page
   - breakthrough distance
   - dwell
   - compensation number depending on those available on the tool.

   The other parameters are optional in this case.



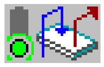
A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion is as follows:
   - Motion at machining feedrate from 1 to 2
   - Dwell for specified duration
   - Retract at retract feedrate from 2 to 3.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page  to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Boring operations:

```
CYCLE/BORE, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT,
           %MFG_CLEAR_TIP, DWELL, %MFG_DWELL_REVOL
```

A typical NC data output is as follows:

```
CYCLE/BORE, 25.000000, 500.000000, MMPM, 5.000000, DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_BORING section of the Manufacturing Infrastructure User's Guide.

Create a Boring Spindle Stop Operation

This task shows how to insert a Boring Spindle Stop operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

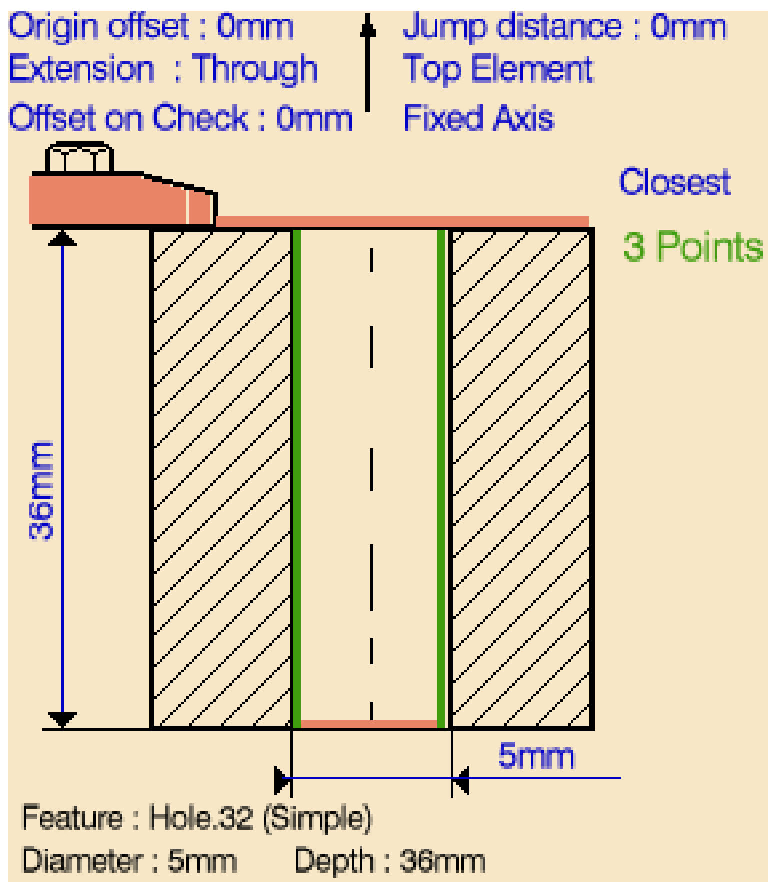
Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the Boring Spindle Stop icon .

   A Boring Spindle Stop entity along with a default tool is added to the program.

   The Boring Spindle Stop dialog box appears directly at the Geometry tab page .

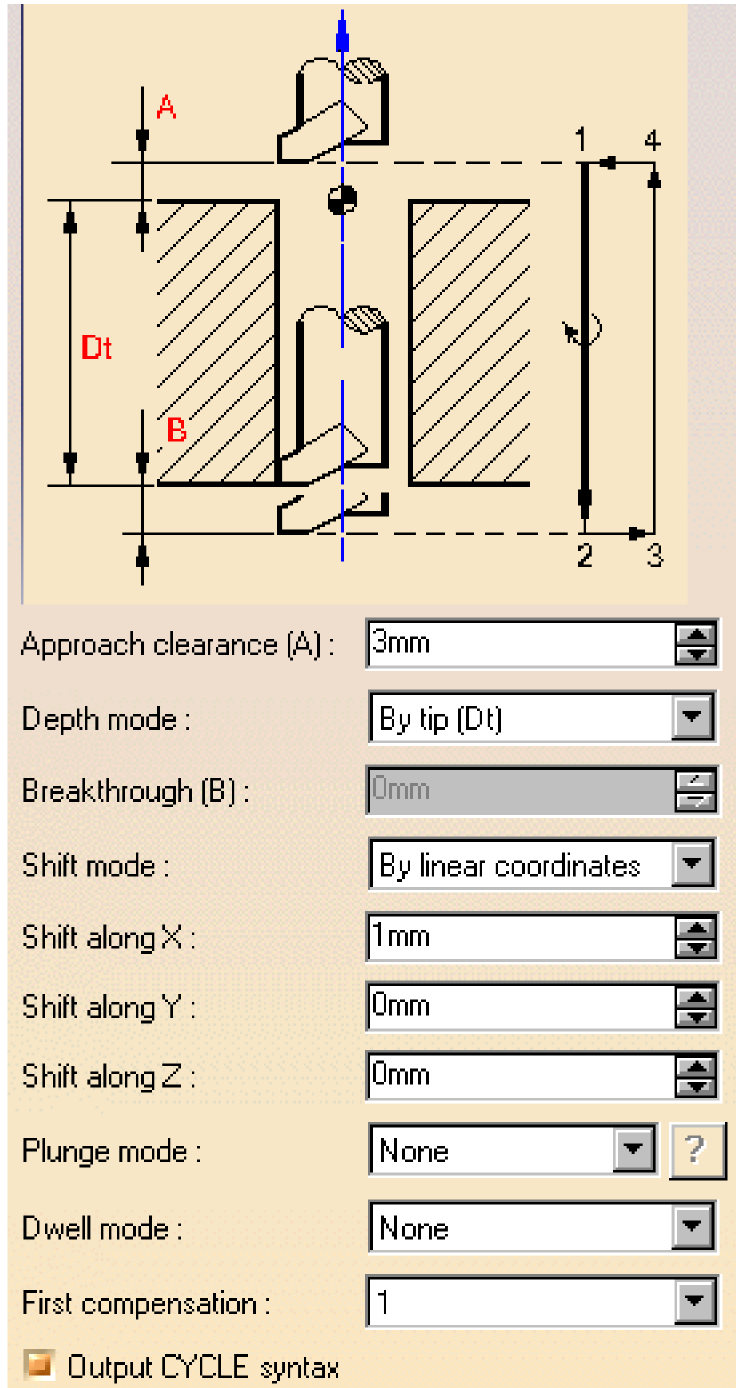


2. Select the red hole depth representation then select the hole geometry in the 3D window.
   Just double click to end your selections.

   The sensitive icon is updated with the following information:
   - depth and diameter of the first selected hole
   - hole extension type: through
   - Number of points to machine.
3. If needed, select the tool axis direction.

4. Select the Strategy tab page to specify the following machining parameters.
   - approach clearance
   - depth mode: by tip
     The depth value used is the one specified in the Geometry tab page.
   - breakthrough distance
   - shift: by linear coordinates (along X)
   - dwell
   - compensation number depending on those available on the tool.

   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion with a boring bar is as follows:

   - Motion at machining feedrate from 1 to 2
   - Dwell for specified duration
   - Spindle stop
   - Shift motion at retract feedrate from 2 to 3
   - Retract at retract feedrate from 3 to 4
   - Shift motion at retract feedrate from 4 to 1.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Boring Spindle Stop operations:

```
CYCLE/BORE, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT,
            %MFG_CLEAR_TIP, ORIENT, %MFG_XOFF, DWELL,
%MFG_DWELL_REVOL
```

A typical NC data output is as follows:

```
CYCLE/BORE, 25.000000, 500.000000, MMPM, 5.000000, ORIENT,
1.000000, DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_BORING__SPINDLE_STOP section of the Manufacturing Infrastructure User's Guide.

Create a Boring and Chamfering Operation

This task shows how to insert a Boring and Chamfering operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

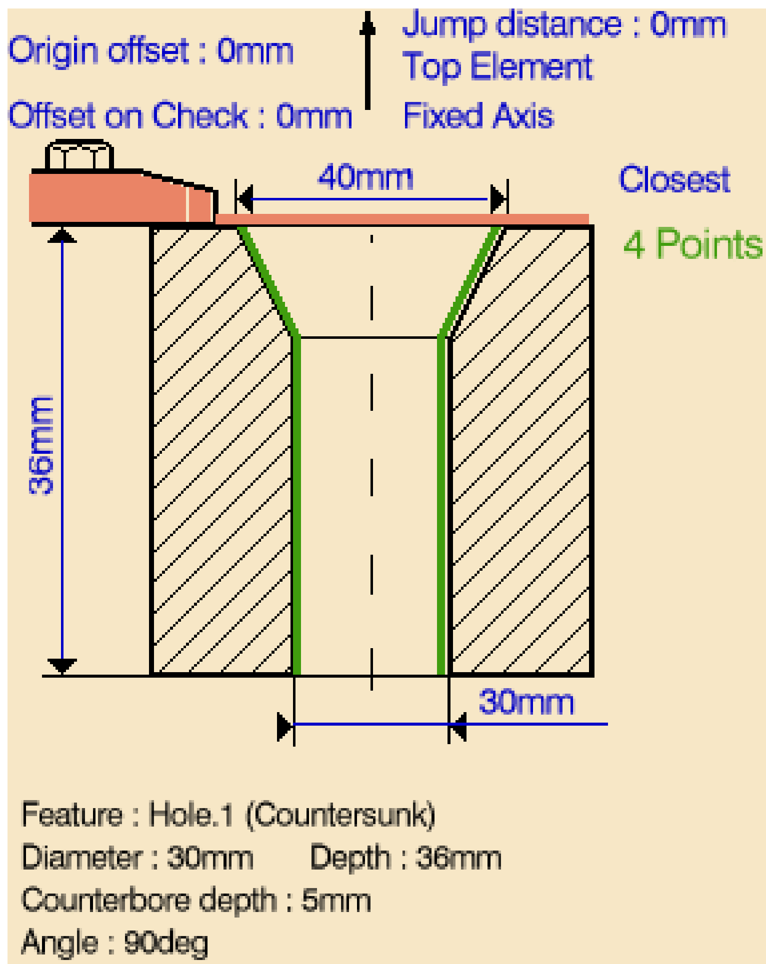
1. Select the Boring and Chamfering icon.

   A Boring and Chamfering entity along with a default tool is added to the program.

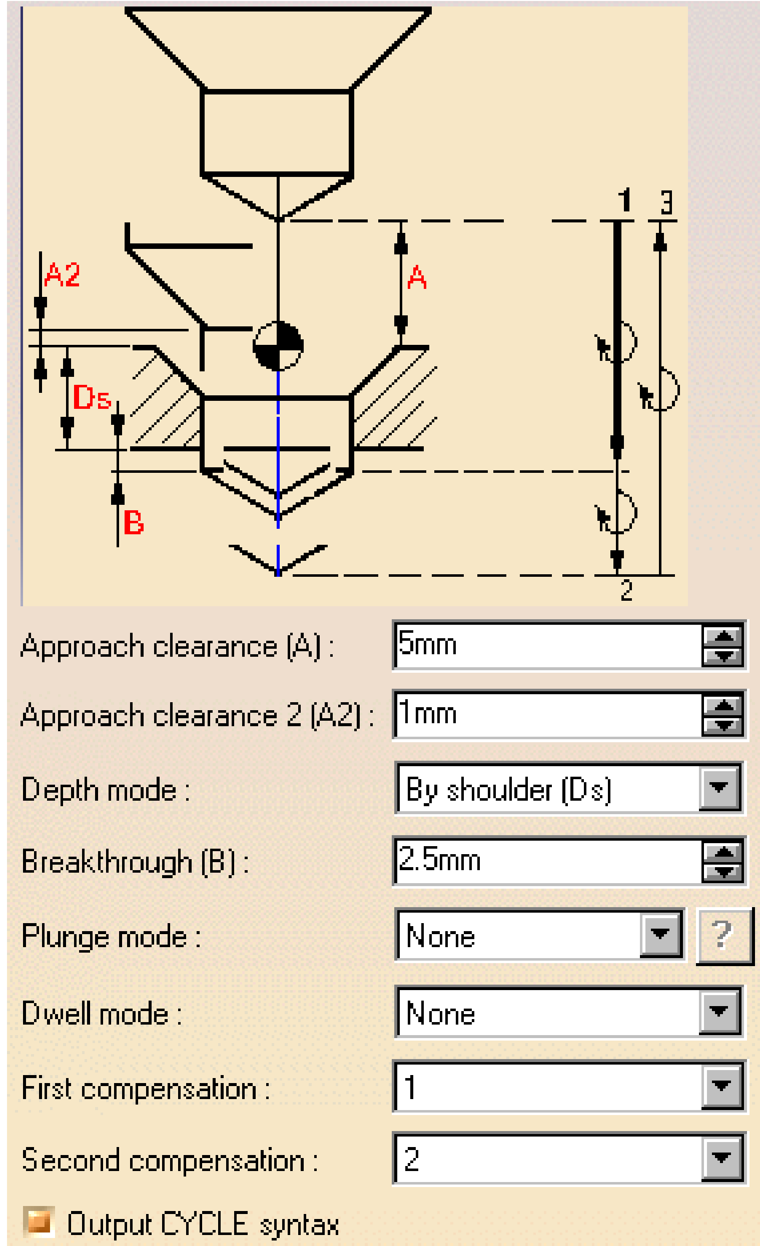
   The Boring and Chamfering dialog box appears directly at the Geometry tab page.
2. Select the red hole depth representation then select hole geometry in the 3D window.
   Just double click to end your selections.

   The sensitive icon is updated with the following information:

   - depth, diameter, conterbore depth and angle of the first selected feature
   - number of points to machine.
3. If needed, select tool axis direction.



4. Select the Strategy tab page to specify the following machining parameters:
   - approach clearances 1 and 2
   - depth mode: by shoulder
     The depth value used is the one specified in the Geometry tab page
   - breakthrough distance
   - dwell
   - first compensation number depending on those available on the tool for boring
   - second compensation number depending on those available on the tool for chamfering.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.
   You can specify a machining feedrate for the boring phase of the operation and a chamfering feedrate for the chamfering phase.
   Similarly, you can specify a machining spindle speed for the boring phase and a smaller spindle speed for the chamfering phase.

Note that in the tool path represented in the strategy page, tool motion is as follows:

Boring

- Motion at machining feedrate from 1 up to the position where hole is to be bored
- Possibly, activation of second tool compensation number
- Rapid feedrate up to a clearance position before start of chamfering.

Chamfering

- Motion at chamfering feedrate from clearance position to 2
- Dwell for specified duration
- Possibly, activation of first tool compensation number
- Retract at retract feedrate from 2 to 3.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Boring and Chamfering operations:

```
CYCLE/BORE, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
%MFG_CHAMFERFEED_VALUE,
            &MFG_FEED_UNIT, %MFG_SPINDLE_MACH_VALUE,
%MFG_SPINDLE_LOW_VALUE,
            &MFG_SPNDL_UNIT, %MFG_CLEAR_TIP, DWELL,
%MFG_DWELL_REVOL
```

A typical NC data output is as follows:

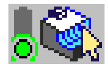
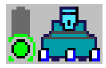
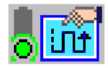
```
CYCLE/BORE, 25.000000, 500.000000, 150.000000, MMPM,
70.000000, 40.000000, RPM, 5.000000, DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_BORING_AND_CHAMFERING section of the Manufacturing Infrastructure User's Guide.

Create a Back Boring Operation

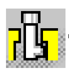
This task shows how to insert a Back Boring operation in the program.

To create the operation you must define:

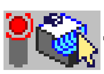
- the geometry of the holes to be machined 
- the tool that will be used 
- the parameters of the machining strategy 
- the feedrates and spindle speeds 
- the macros (transition paths) .

Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the Back Boring icon .

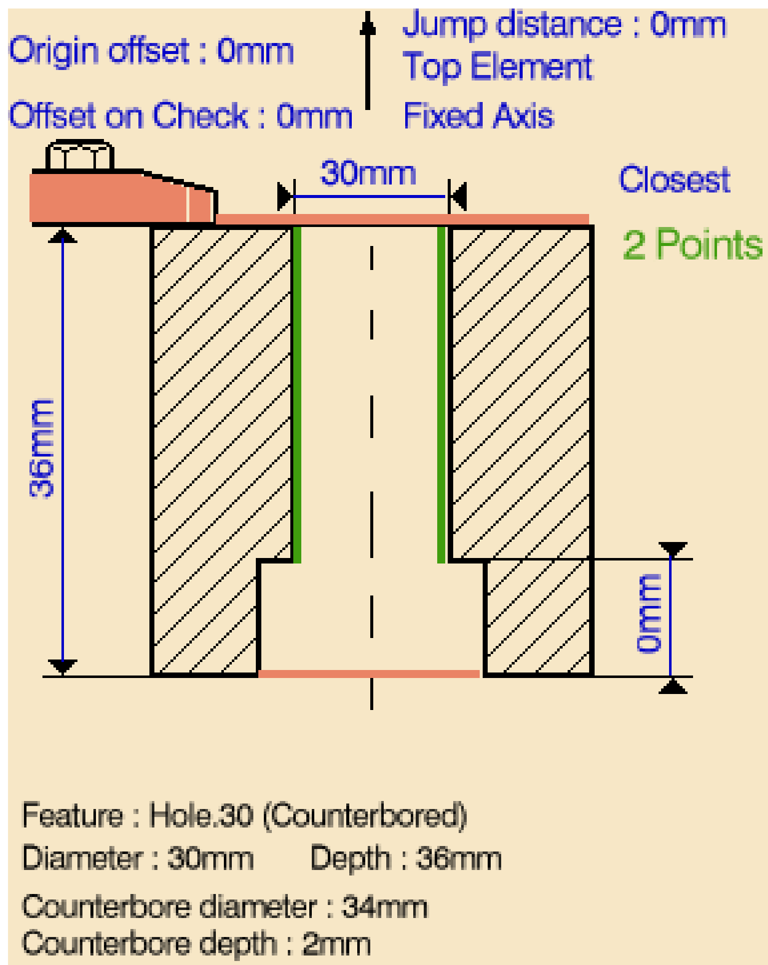
   A Back Boring entity along with a default tool is added to the program.

   The Back Boring dialog box appears directly at the Geometry tab page .

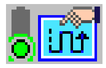
2. Select the top plane representation then select the top of the part.
3. Select the red hole depth representation then specify the hole pattern to be machined by selecting the two counterbored features in the 3D window.

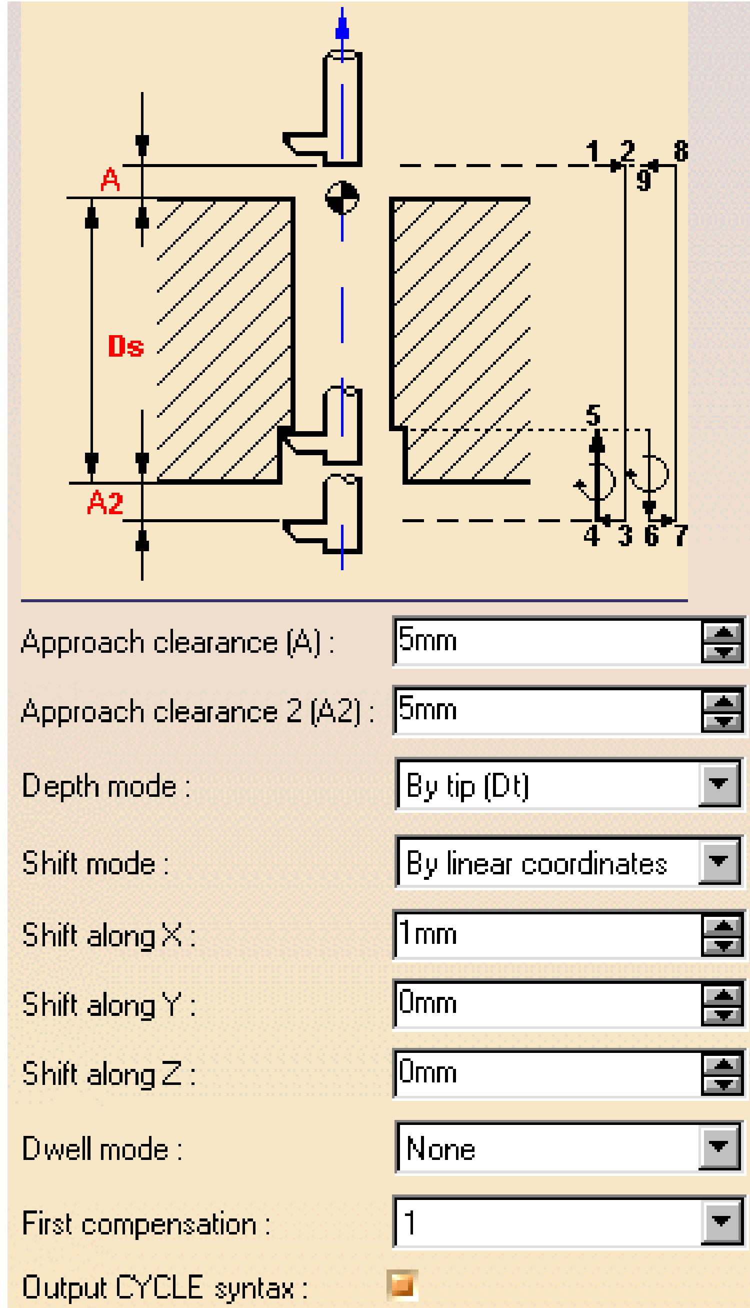
   Just double click to end your selections.

   The Geometry page is updated with information about the first selected feature.

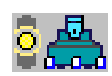


4. If needed, select the tool axis direction.

5. Select the Strategy tab page  to specify the following machining parameters.
   - approach clearances A and A2
   - depth mode: by tip
     The depth value used is the one specified in the Geometry tab page
   - shift: by linear coordinates (along X)
   - dwell
   - compensation number depending on those available on the tool.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

6. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion is as follows:

   - Shift motion at rapid feedrate from 1 to 2
   - Motion at rapid feedrate from 2 to 3
   - Shift motion at rapid feedrate from 3 to 4
   - Motion at machining feedrate from 4 to 5
   - Dwell for specified duration
   - Motion at retract feedrate from 5 to 6
   - Shift motion at retract feedrate from 6 to 7
   - Retract at retract feedrate from 7 to 8
   - Shift motion at retract feedrate from 8 to 9.

7. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

8. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Back Boring operations:

```
CYCLE/BORE, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE, &MFG_FEED_UNIT,
            %MFG_CLEAR_TIP, ORIENT, %MFG_XOFF, DWELL, %MFG_DWELL_REVOL
```

A typical NC data output is as follows:

```
CYCLE/BORE, 25.000000, 500.000000, MMPM, 5.000000, ORIENT, 1.000000, DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_BACK_BORING section of the Manufacturing Infrastructure User's Guide.

Create a Tapping Operation

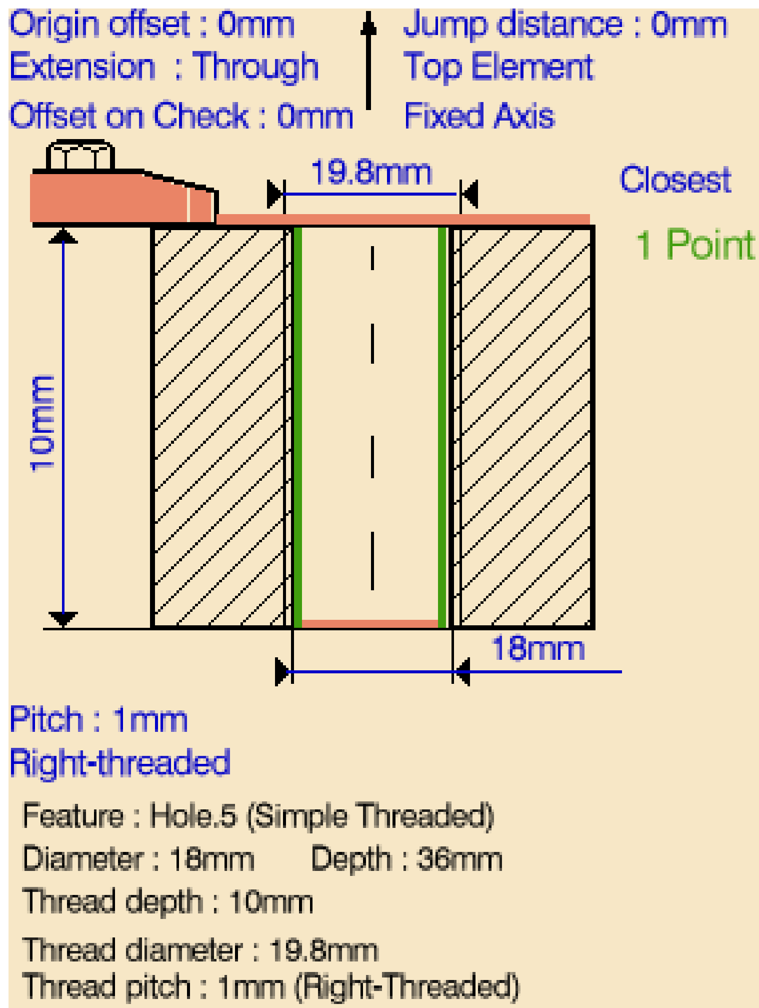
This task shows how to insert a Tapping operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.



1. Select the Tapping icon.

   A Tapping entity along with a default tool is added to the program.

   The Tapping dialog box appears directly at the Geometry tab page.

   This tab page includes an icon representing a simple hole. There are several hot spots in the icon.
2. Select the red hole depth representation then select a threaded hole feature in the 3D window.
   Just double click to end your selection.

   The sensitive icon is updated with the following:

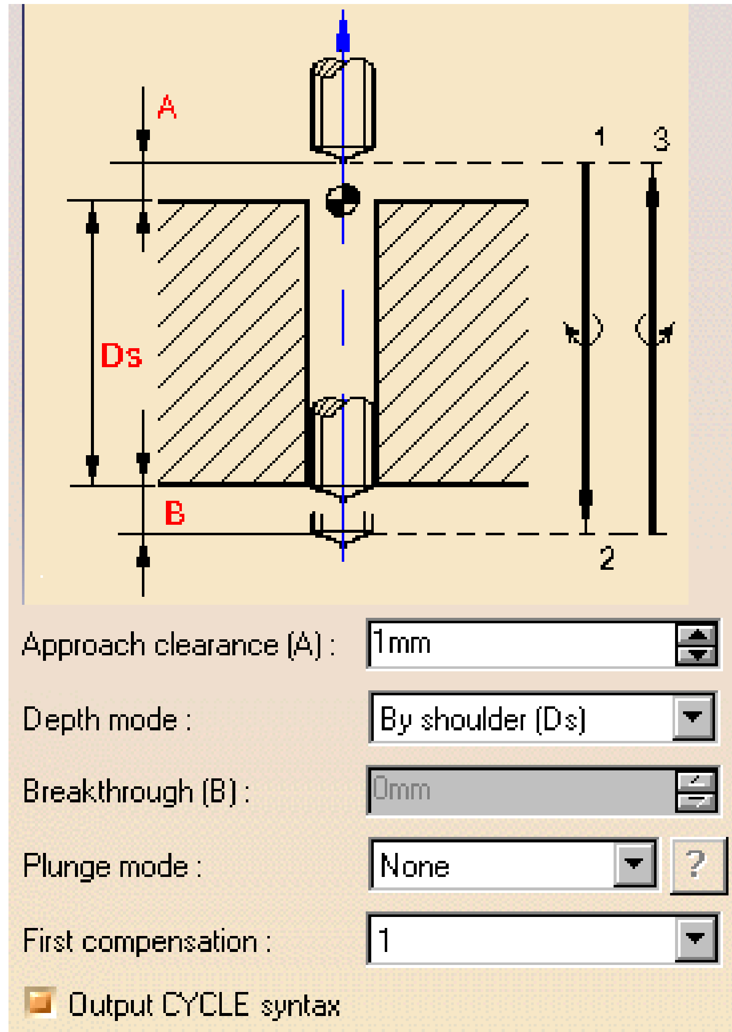
   - thread depth and thread diameter
   - hole extension type
   - thread pitch
   - thread direction.

   You can modify this data.
   Other values are shown for information

only.

3. If needed, select tool axis direction.
4. Select the Strategy tab page to specify the following machining parameters.
   - approach clearance
   - depth mode: by shoulder
     The depth value used is the one specified in the Geometry tab page.
   - compensation number depending on those available on the tool.

   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion is as follows:
   - Motion at machining feedrate from 1 to 2
   - Reverse spindle rotation
   - Retract at machining feedrate from 2 to 3
   - Reverse spindle rotation.

If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

The general procedure for this is described in Define Macros of an Axial Machining Operation.

Before accepting the operation, you should check its validity by replaying the tool path.

6. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Tapping operations:

```
CYCLE/TAP, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT, %MFG_CLEAR_TIP
```

A typical NC data output is as follows:

```
CYCLE/TAP, 38.500000, 500.000000, MMPM, 2.500000
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_TAPPING section of the Manufacturing Infrastructure User's Guide.

Create a Reverse Threading Operation

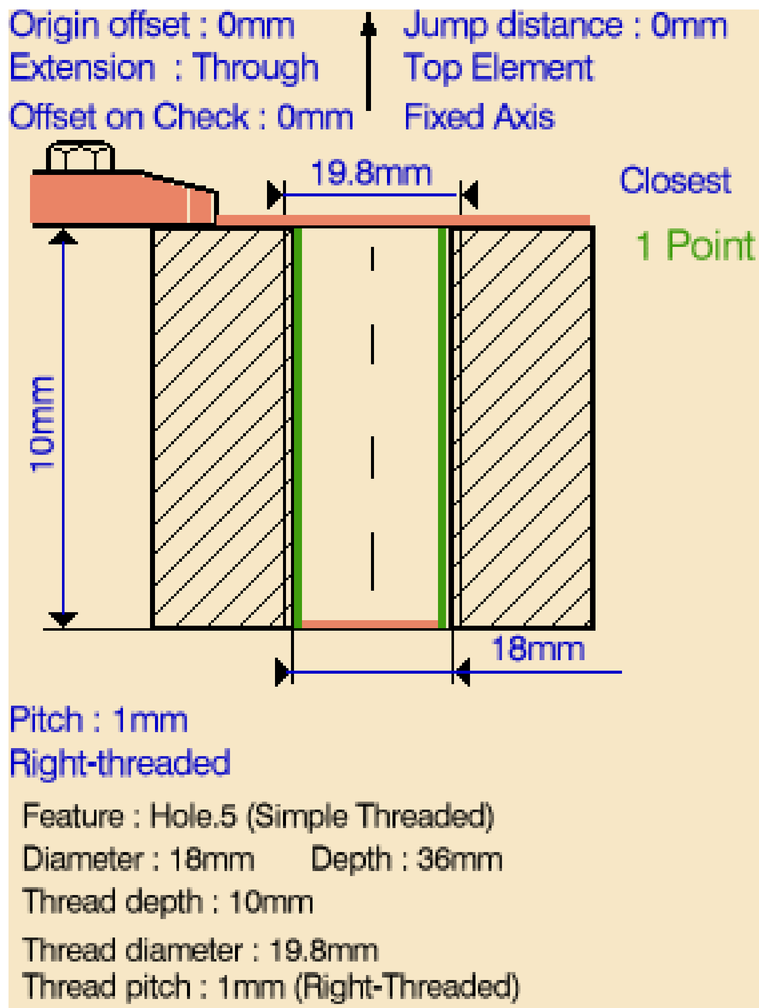
This task shows how to insert a Reverse Threading operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.



1. Select the Reverse Threading icon .

   A Reverse Threading entity along with a default tool is added to the program.

   The Reverse Threading dialog box appears directly at the Geometry tab page .

2. Select the red hole depth representation then select a threaded hole feature in the 3D window.
   Just double click to end your selection.

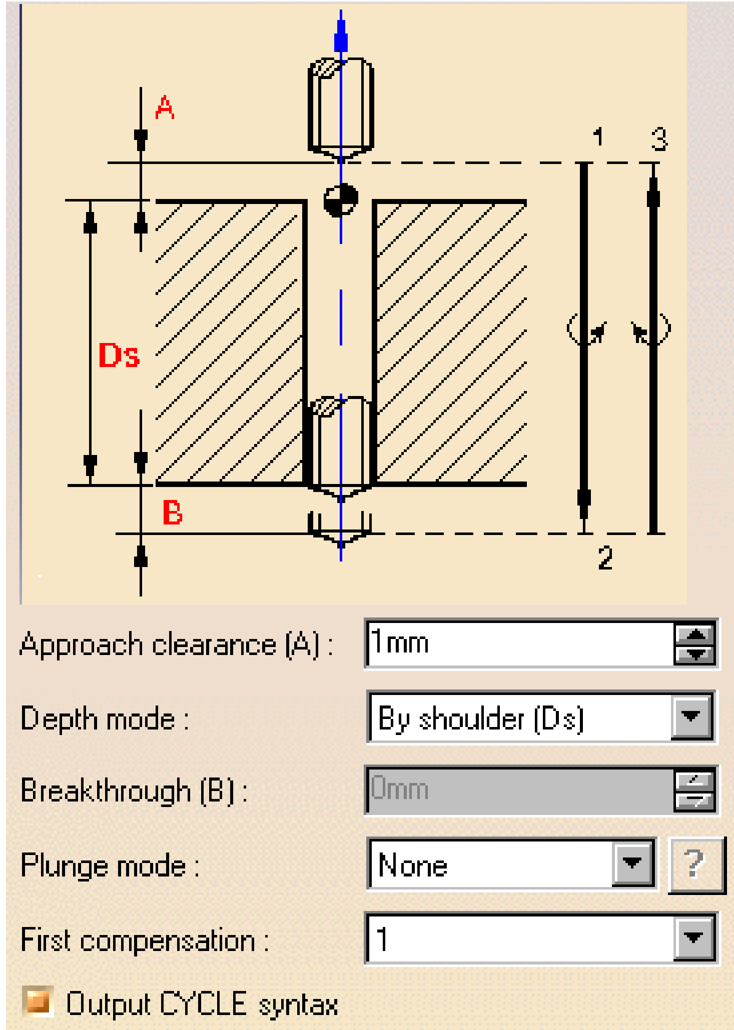
   The sensitive icon is updated with the following:

   - thread depth and thread diameter
   - hole extension type
   - thread pitch
   - thread direction.

   You can modify this data.
   Other values are shown for information only.

3. If needed, select the tool axis direction.
4. Select the Strategy tab page to specify the following machining parameters.
   - approach clearance
   - depth mode: by shoulder
     The depth value used is the one specified in the Geometry tab page.
   - compensation number depending on those available on the tool.

   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion is at:
   - Motion at machining feedrate from 1 to 2
   - Spindle off then reverse spindle rotation
   - Retract at machining feedrate from 2 to 3.
6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Reverse Threading operations:

```
CYCLE/TAP, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT, %MFG_CLEAR_TIP
```

A typical NC data output is as follows:

```
CYCLE/TAP, 38.500000, 500.000000, MMPM, 2.500000
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_REVERSE_THREADING section of the Manufacturing Infrastructure User's Guide.

Create a Thread without Tap Head Operation

This task shows how to insert a Thread without Tap Head operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the Thread without Tap Head icon.

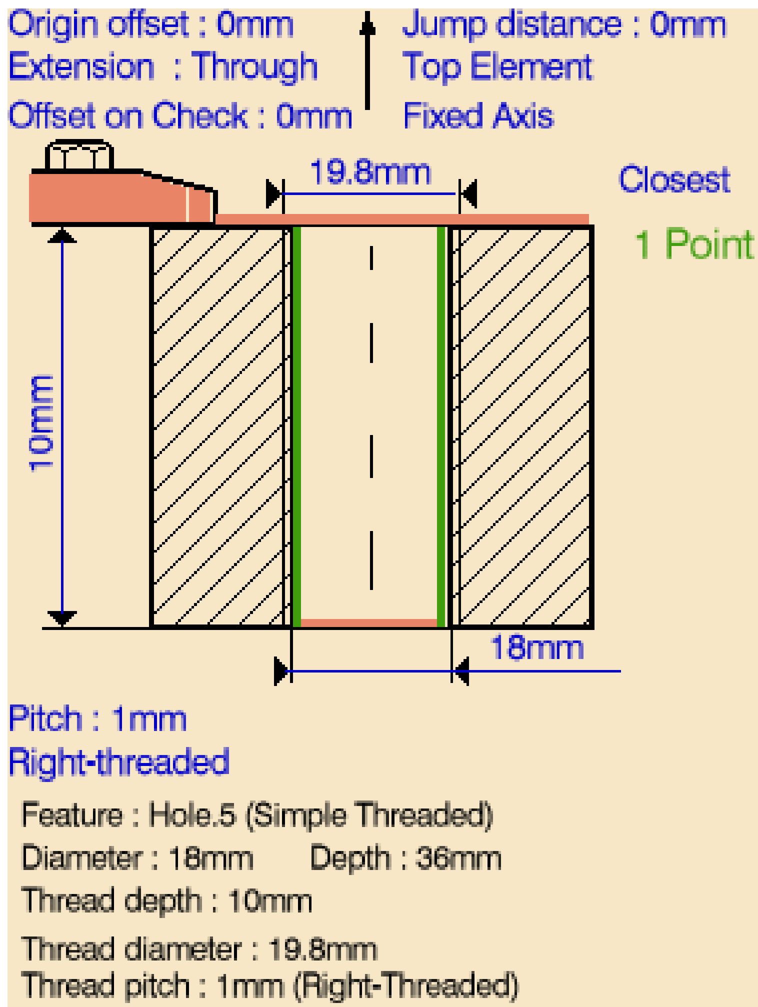
   A Thread without Tap Head entity along with a default tool is added to the program.

   The Thread without Tap Head dialog box appears directly at the Geometry tab page.

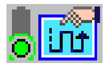
2. Select the red hole depth representation then select a threaded hole feature in the 3D window.
   Just double click to end your selection.

   The sensitive icon is updated with the following:

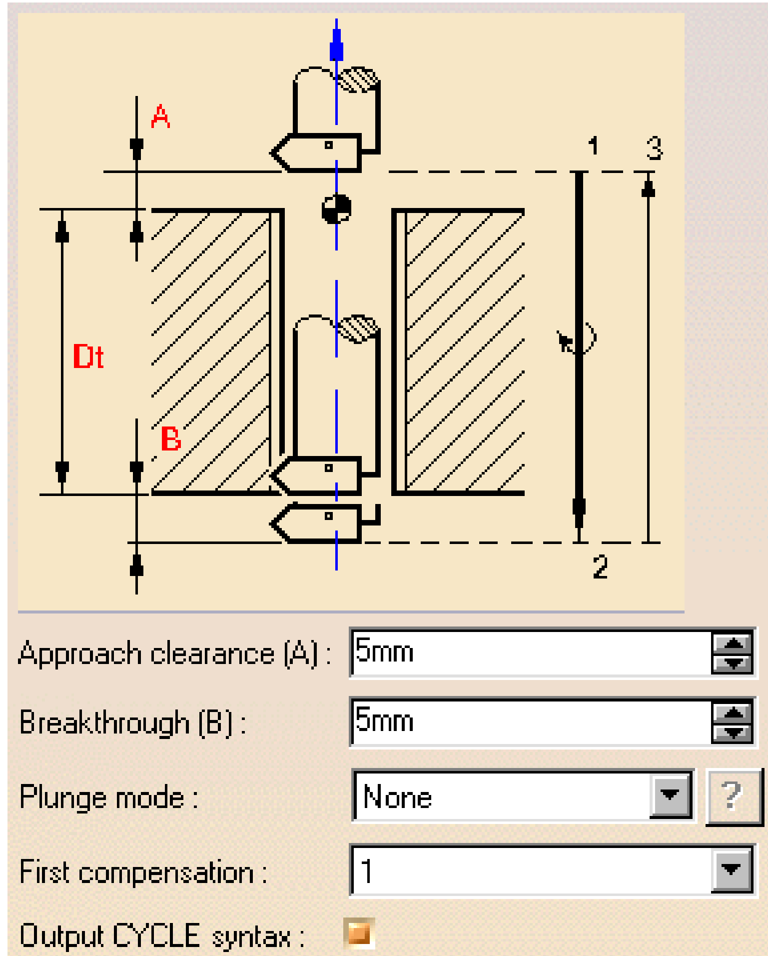
   - thread depth and thread diameter
   - hole extension type
   - thread pitch
   - thread direction.



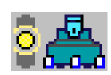
You can modify this data.
Other values are shown for information only.

3. If needed, select the tool axis direction.
4. Select the Strategy tab page  and specify the following machining parameters.
   - approach clearance
   - breakthrough
   - compensation number depending on those available on the tool.

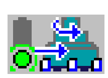
   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

   Note that in the toolpath represented in the strategy page, tool motion is as follows:
   - Motion at machining feedrate from 1 to 2
   - Spindle stop
   - Retract at retract feedrate from 2 to 3.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page  to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Thread without Tap Head operations:

```
CYCLE/TAP, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT, %MFG_CLEAR_TIP
```

A typical NC data output is as follows:

```
CYCLE/TAP, 38.500000, 500.000000, MMPM, 2.500000
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_THREAD_WITHOUT_TAP_HEAD section of the Manufacturing Infrastructure User's Guide.

Create a Thread Milling Operation

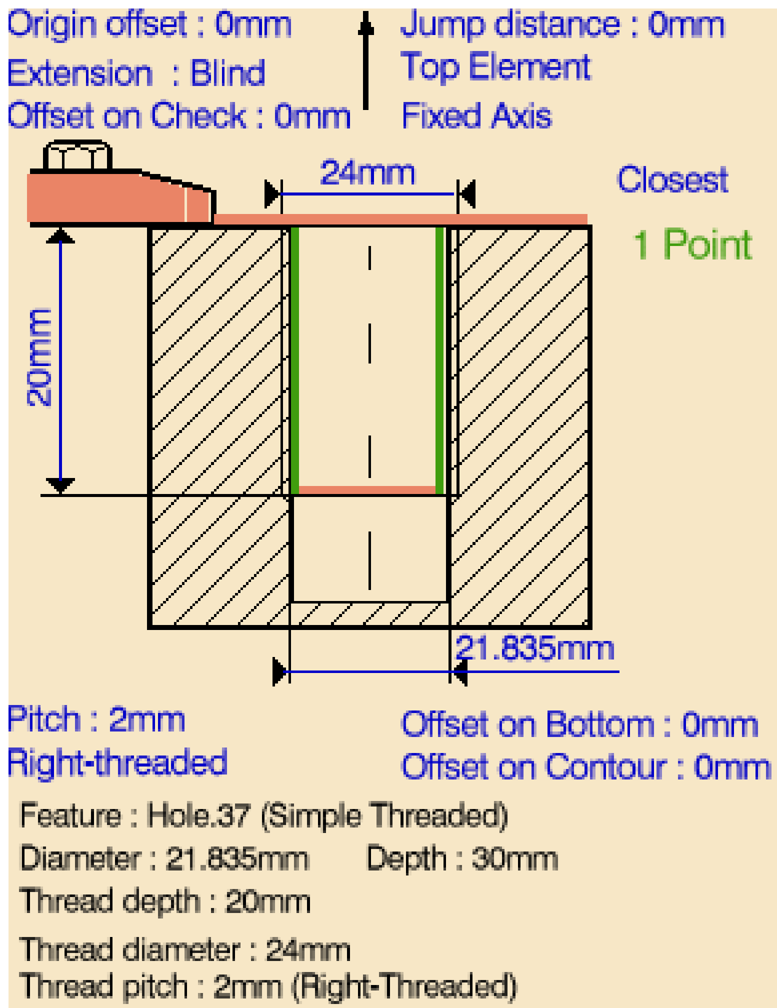
This task shows how to insert a Thread Milling operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.



1. Select the Thread Milling icon .

   A Thread Milling entity along with a default tool is added to the program.

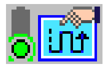
   The Thread Milling dialog box appears directly at the Geometry tab page .

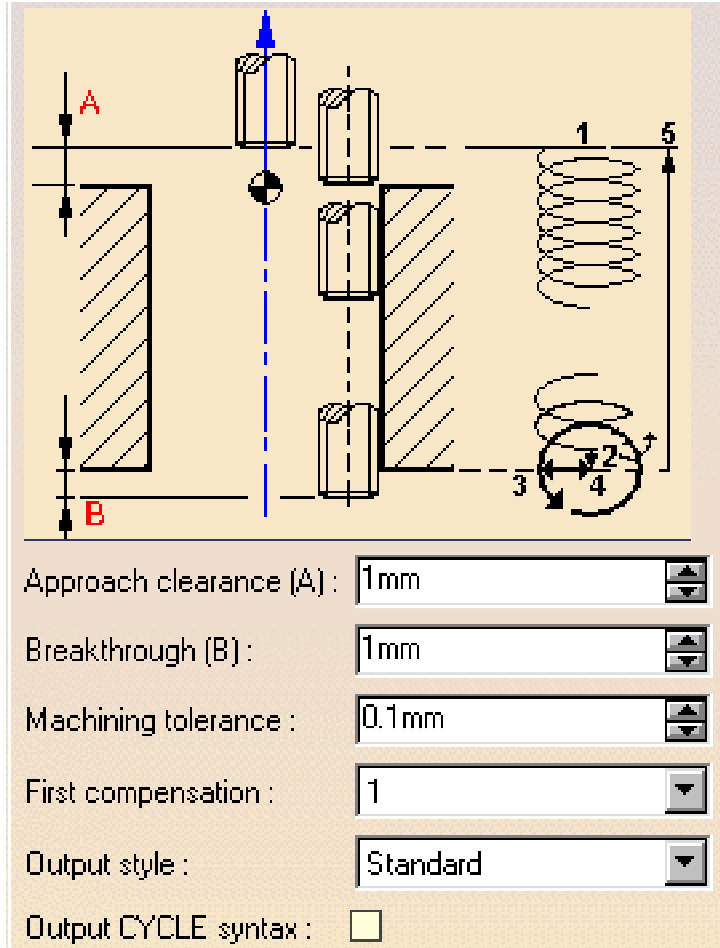
2. Select the red hole depth representation then select a threaded hole feature in the 3D window.
   Just double click to end your selection.

   The sensitive icon is updated with the following:

   - thread depth and thread diameter
   - hole extension type
   - thread pitch
   - thread direction.

   You can modify this data.
   Other values are shown for information only.

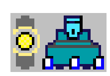
3. If needed, enter offset values for the Bottom and Contour.
4. If needed, select the tool axis direction.
5. Select the Strategy tab page  and set the machining parameters:
   - Approach clearance
   - Breakthrough
   - Machining tolerance
   - Compensation number depending on those available on the tool.
   - Output style: standard tip or cutter profile.

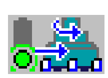
   If you want to generate CYCLE statements, you must select the Output CYCLE syntax checkbox **and** set the Syntax Used option to Yes in the NC Output generation dialog box. Otherwise, GOTO statements will be generated.

   If a cutter profile output style is selected, both the tip and cutter profile will be visualized during tool path replay.
   The cutter profile output allows easier tool compensation to be done on the shop floor.

A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

6. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

   Note that in the toolpath represented in the strategy page, tool motion is at:
   - Motion at machining feedrate from 1 to 2
   - Motion at feedrates defined on macros from 2 to 3 and 3 to 4
   - Retract at retract feedrate from 4 to 5.

7. Select the Macros tab page to specify the operation's transition paths (approach and retract motion, for example).

   The general procedure for this is described in Define Macros of an Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

8. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Thread Milling operations:

```
CYCLE/TAP, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT, %MFG_CLEAR_TIP
```

A typical NC data output is as follows:

```
CYCLE/TAP, 38.500000, 500.000000, MMPM, 2.500000
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_THREAD_MILLING section of the Manufacturing Infrastructure User's Guide.

Create a Countersinking Operation

This task shows how to insert a Countersinking operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

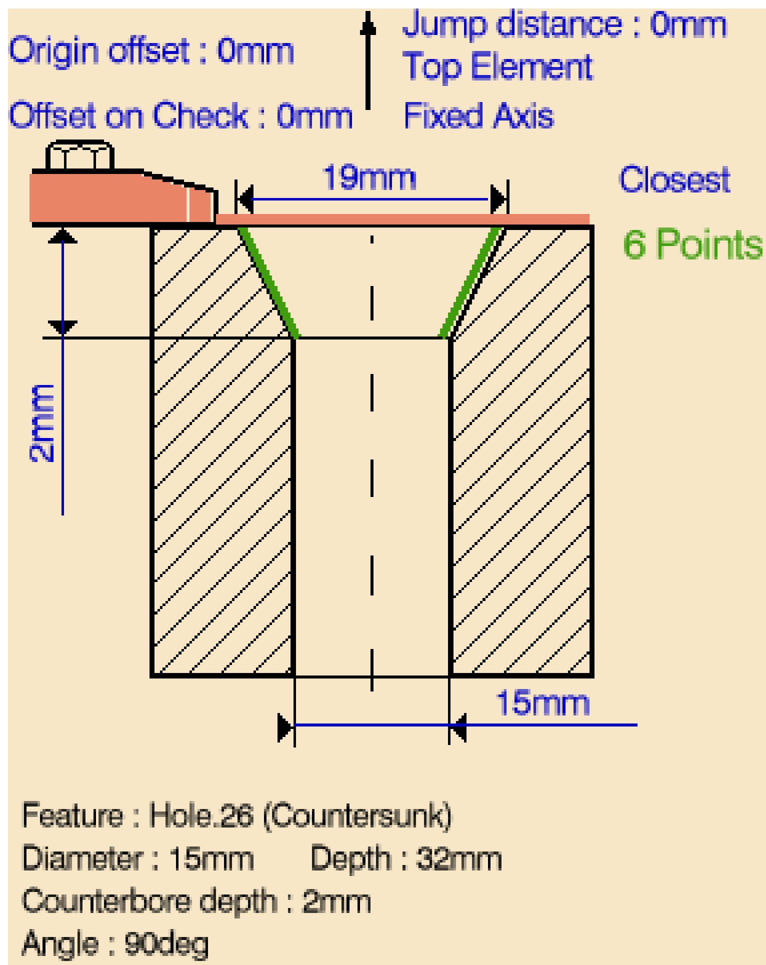
Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the Countersinking icon .

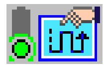
   A Countersinking entity along with a default tool is added to the program.

   The Countersinking dialog box appears directly at the Geometry tab page .

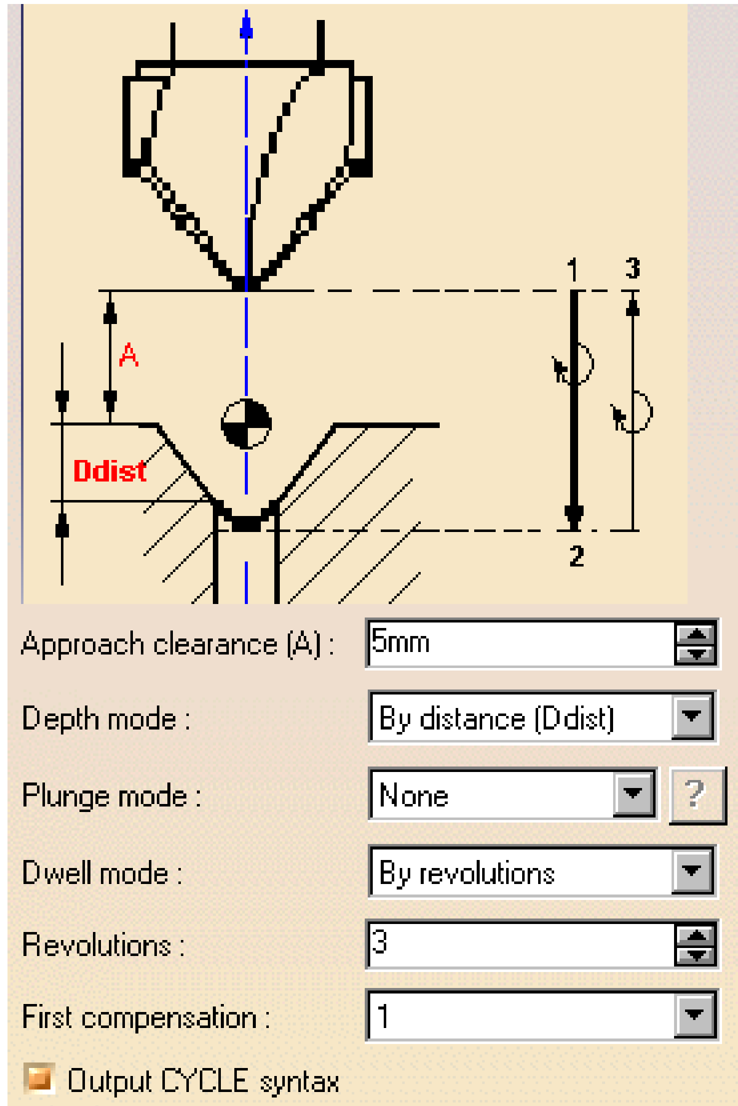


2. Select the red hole depth representation then select hole geometry in the 3D window. Just double click to end your selections.

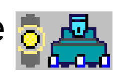
3. If needed, select the tool axis direction.

4. Select the Strategy tab page  to specify the following machining parameters.
   - Approach clearance
   - Depth mode: by distance
     The depth value used is the one specified in the Geometry tab page.
   - Dwell
   - Compensation number depending on those available on the tool.

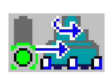
   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

   Note that in the toolpath represented in the strategy page, tool motion is at:
   - Motion at machining feedrate from 1 to 2
   - Dwell for specified duration
   - Increment at finishing feedrate from 2 to 3
   - Retract at retract feedrate from 3 to 4.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Countersinking operations:

```
CYCLE/CSINK, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT,
             %MFG_CLEAR_TIP, DWELL, %MFG_DWELL_REVOL
```

A typical NC data output is as follows:

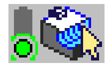
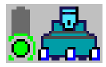
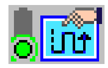
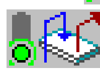
```
CYCLE/CSINK, 25.000000, 500.000000, MMPM, 5.000000, DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_COUNTERSINKING section of the Manufacturing Infrastructure User's Guide.

Create a Chamfering Two Sides Operation

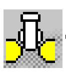
This task shows how to insert a Chamfering Two Sides operation in the program.

To create the operation you must define:

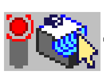
- the geometry of the holes to be machined 
- the tool that will be used 
- the parameters of the machining strategy 
- the feedrates and spindle speeds 
- the macros (transition paths) .

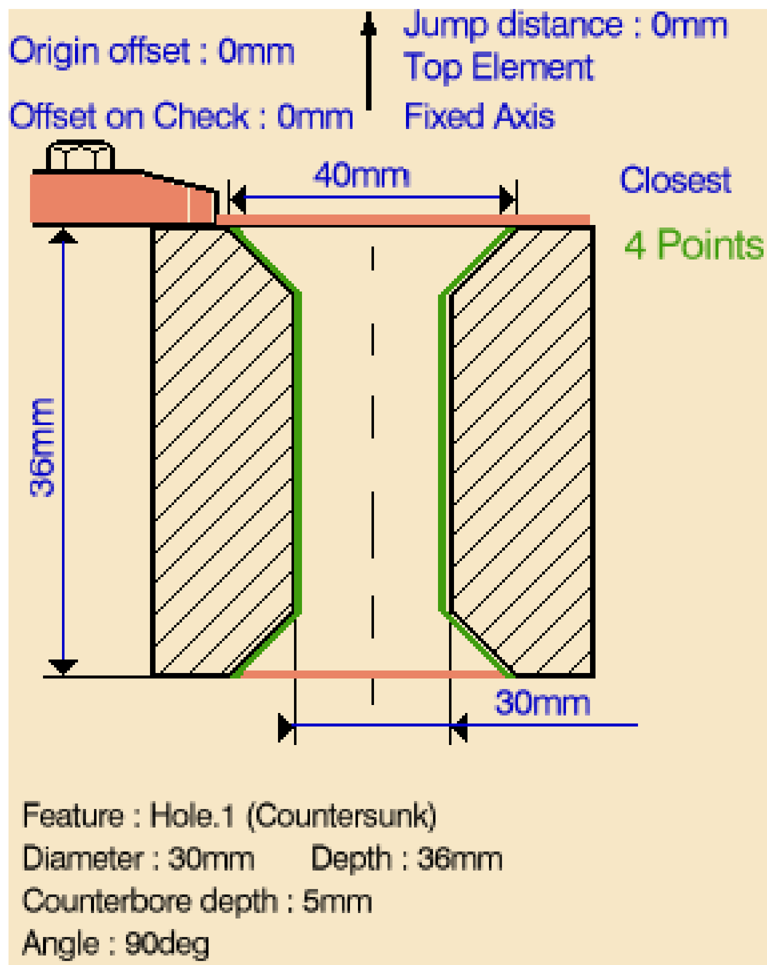
Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the Chamfering Two Sides icon .

   A Chamfering Two Sides entity along with a default tool is added to the program.

   The Chamfering Two Sides dialog box appears directly at the Geometry tab page .

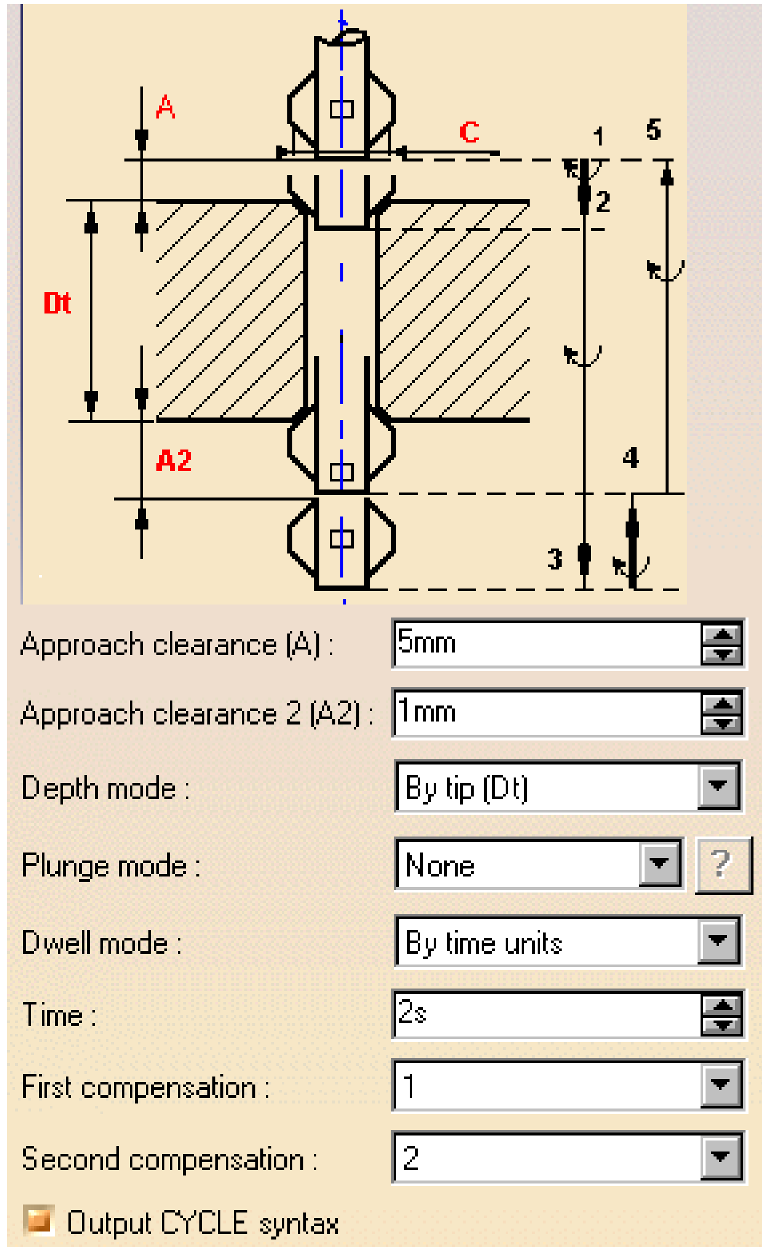


2. Select the red hole depth representation then select the hole geometry in the 3D window.
   Just double click to end your selections.

3. If needed, select the tool axis direction.

4. Select the Strategy tab page to specify the following machining parameters:
   - approach clearances 1 and 2
   - depth mode: by tip
   - breakthrough distance
   - dwell in seconds
   - first compensation number depending on those available on the tool for top chamfering
   - second compensation number depending on those available on the tool for bottom chamfering.

   Please note that the depth value and chamfer diameter are retrieved from your geometry selections.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the tool path represented in the strategy page, tool motion is as follows:

   - Motion at machining feedrate from 1 to 2
   - Dwell for specified duration
   - Possibly, activation of second tool compensation number (output point change)
   - Motion at approach feedrate from 2 to 3
   - Motion at machining feedrate from 3 to 4
   - Dwell for specified duration
   - Possibly, activation of first tool compensation number (output point change)
   - Retract at retract feedrate from 4 to 5.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Chamfering Two Sides operations:

```
CYCLE/BORE, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE, &MFG_FEED_UNIT,
            %MFG_CLEAR_TIP, DWELL, %MFG_DWELL_REVOL
```

A typical NC data output is as follows:

```
CYCLE/BORE, 25.000000, 500.000000, MMPM, 5.000000, DWELL, 3
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_TWO_SIDES_CHAMFERING section of the Manufacturing Infrastructure User's Guide.

Create a T-Slotting Operation

This task shows how to insert a T-Slotting operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

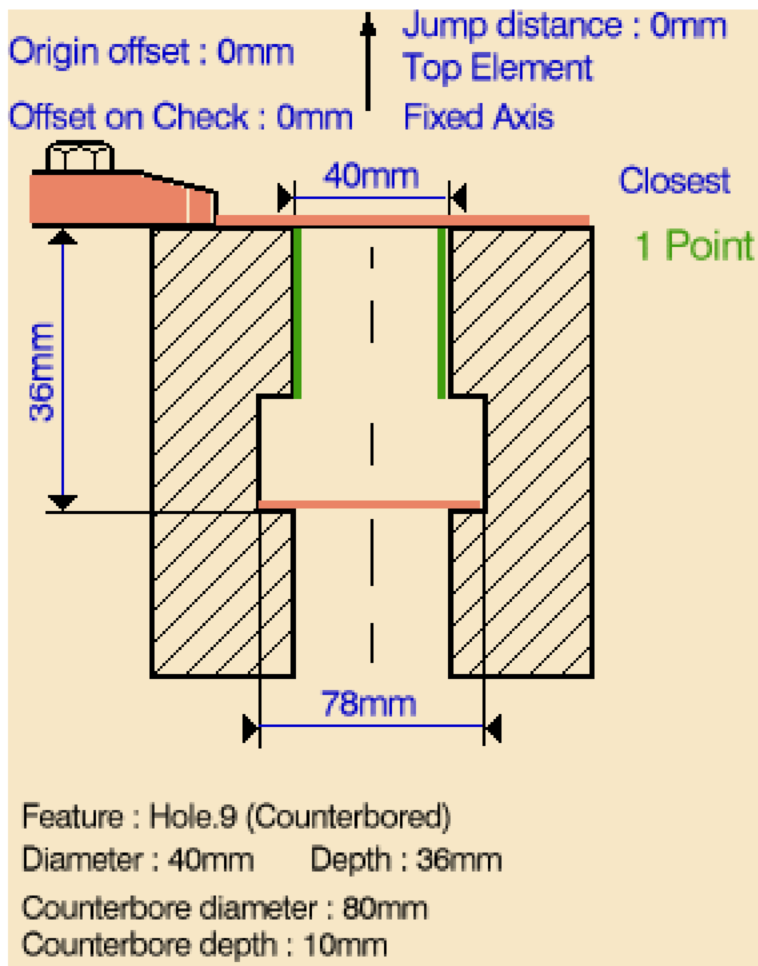
Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.

1. Select the T-Slotting icon .

   A T-Slotting entity along with a default tool is added to the program.

   The T-Slotting dialog box appears directly at the Geometry tab page .



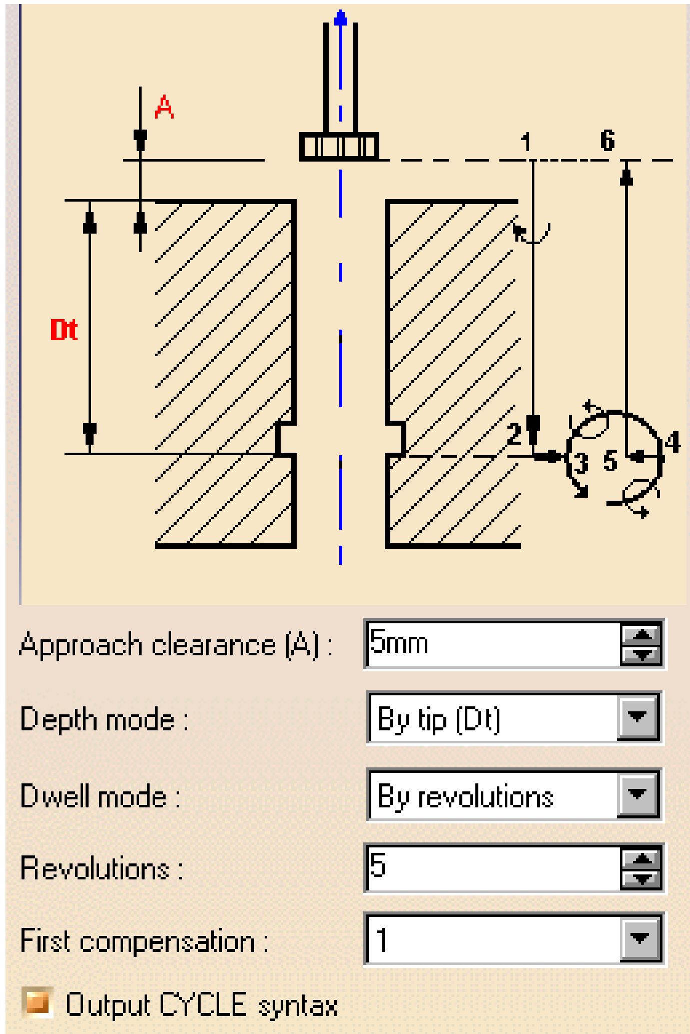
2. Select the red hole depth representation then select the desired hole geometry in the 3D window. Just double click to end your selections.

   The sensitive icon is updated the depth and diameter of the selected feature.
   In this example, the slot diameter (78mm) was entered manually

3. If needed, select the tool axis direction.

4. Select the Strategy tab page to specify the following machining parameters.
    - Approach clearance
    - Depth mode: by tip
      The depth value used is the one specified in the Geometry tab page.
    - Dwell
    - Compensation number depending on those available on the tool.

   The other parameters are optional in this case.



A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use.

This is described in Edit the Tool of an Operation.

5. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the toolpath represented in the strategy page, tool motion is at:
    - Motion at approach feedrate from 1 to 2
    - Motion at machining feedrate from 2 to 3
    - Retract at retract feedrate from 3 to 4.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page to specify the desired transition paths.

   The general procedure for this is described in Define Macros of an Axial Machining Operation.

Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for T-Slotting operations:

```
CYCLE/TAP, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT, %MFG_CLEAR_TIP
```

A typical NC data output is as follows:

```
CYCLE/TAP, 38.500000, 500.000000, MMPM, 2.500000
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_T_SLOTTING section of the Manufacturing Infrastructure User's Guide.

Create a Circular Milling Operation

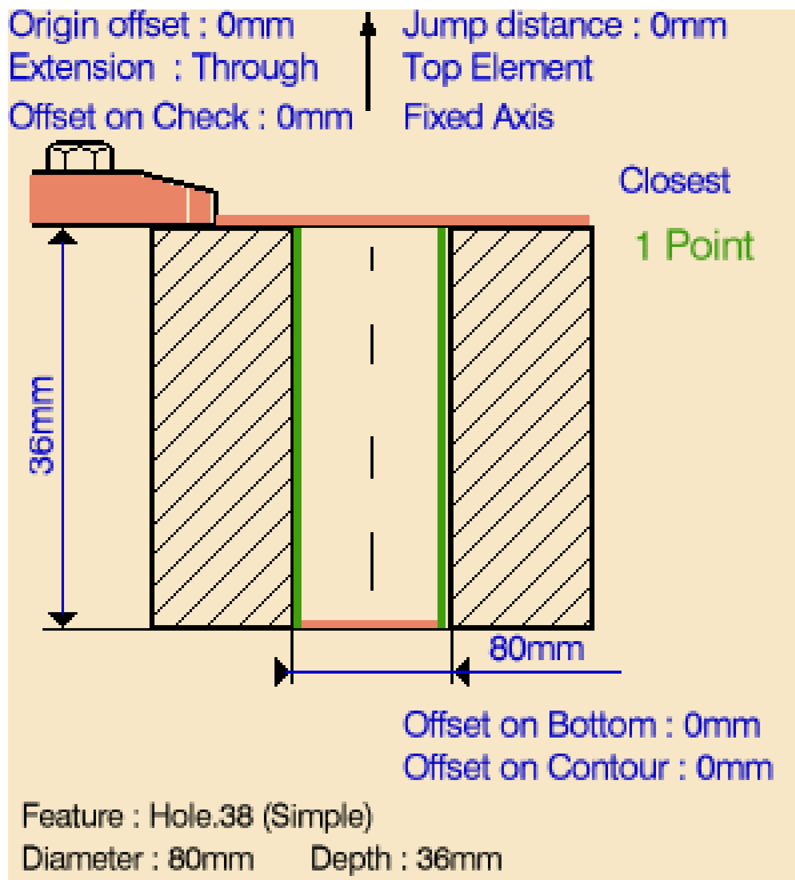
This task shows how to insert a Circular Milling operation in the program.

To create the operation you must define:

- the geometry of the holes to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the HoleMakingOperations.CATPart document, then select the desired NC Manufacturing workbench from the Start menu.

Make the Manufacturing Program current in the specification tree.



1. Select the Circular Milling icon .

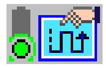
   A Circular Milling entity along with a default tool is added to the program.

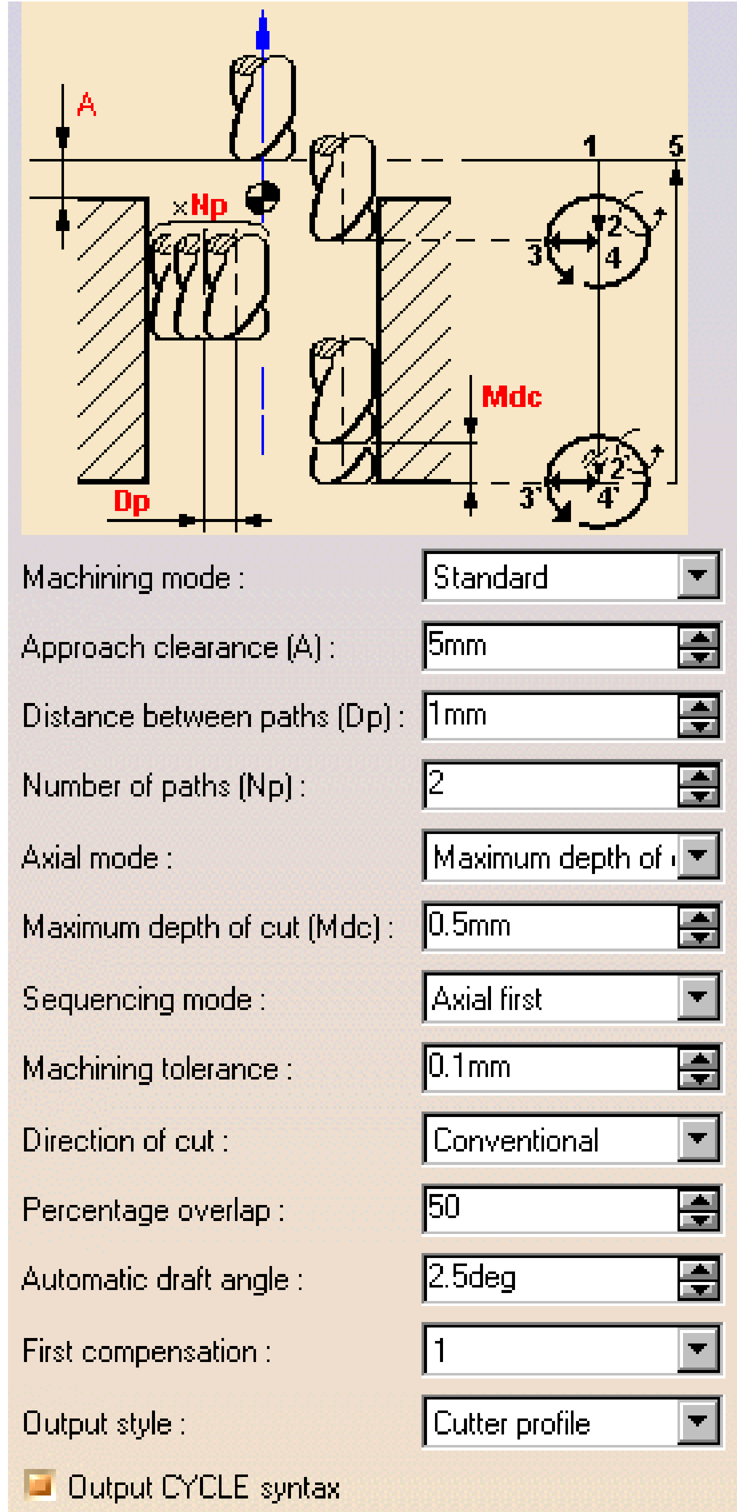
   The Circular Milling dialog box appears directly at the Geometry tab page .

2. If needed, enter Offset values for the Bottom and Contour.

3. Select the red hole depth representation then select hole geometry in the 3D window. Just double click to end your selections.

4. If needed, select the tool axis direction.

5. Select the Strategy tab page  and choose the machining mode:
   - Standard
   - Helical.



6. Specify the machining parameters. The following are common to the two machining modes:
   - Approach clearance
   - Machining tolerance
   - Direction of cut
   - Compensation number depending on those available on the tool
   - Output style: standard tip or cutter profile.

If a cutter profile output style is selected, both the tip and cutter profile will be visualized during tool path replay.
The cutter profile output allows easier tool compensation to be done on the shop floor.

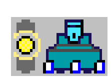
Standard machining parameters:

- Number of paths and distance between paths
- Axial mode:
  Maximum depth of cut
  or Number of levels
- Sequencing mode:
  Axial first or Radial first
- Percentage overlap
- Automatic draft angle.

Helical machining parameters:

- Helix mode: By Angle or By Pitch
- Angle or Pitch value.

A tool is proposed by default when you want to create a machining operation.

If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use. This is described in Edit the Tool of an Operation.

7. Select the Feeds and Speeds tab page to specify the feedrates and spindle speeds for the operation.

   Note that in the toolpath represented in the strategy page, tool motion is at:

   - Motion at machining feedrate from 1 to 2
   - Motion at feedrates defined on macros from 2 to 3, 3 to 4, 4 to 2', 2' to 3' and 3' to 4'
   - Retract at retract feedrate from 4' to 5.

8. Select the Macros tab page to specify the operation's transition paths (approach and retract motion, for example).

   The general procedure for this is described in Define Macros of an Operation.

   Before accepting the operation, you should check its validity by replaying the tool path.

9. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Circular Milling operations:

```
CYCLE/CIRCULARMILLING, %MFG_TOTAL_DEPTH, %MFG_FEED_MACH_VALUE,
&MFG_FEED_UNIT,
                       %MFG_CLEAR_TIP
```

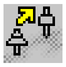
A typical NC data output is as follows:

```
CYCLE/CIRCULARMILLING, 38.500000, 500.000000, MMPM, 2.500000
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_CIRCULAR_MILLING section of the Manufacturing Infrastructure User's Guide.

Auxiliary Operations

The tasks for inserting auxiliary operations in the manufacturing program are documented in the *NC Manufacturing Infrastructure User's Guide*.


Insert Tool Change: Select the Tool Change icon then select the tool type to be referenced in the tool change.


Insert Machine Rotation: Select the Machine Rotation icon then specify the tool rotation characteristics.


Insert Machining Axis System or Origin: Select the Machining Axis or Origin icon then specify the characteristics of the machining axis system or origin.


Insert PP Instruction: Select the PP Instruction icon then enter the syntax of the PP instruction.


Insert Copy Transformation Instruction **(P2 functionality)**: Select the Copy Transformation icon then select the reference operation. You can then specify the number of copies and the characteristics of the transformation.

Part Operations, Manufacturing Programs and Machining Processes

The following task for optimizing the order of operations in a program according to pre-defined sequencing rules is described in this guide:

Auto-sequence Operations in a Program **(P2 functionality)**: Verify the administrator's settings for sequencing rules and priorities. If you are authorized, you can adjust these settings before applying the Auto-sequencing to your program.

The tasks for creating and managing Part Operations, Manufacturing Programs and Machining Processes are documented in the *NC Manufacturing Infrastructure User's Guide*.

Create and Edit a Part Operation: Select the Part Operation icon then specify the entities to be referenced by the part operation: machine tool, machining axis system, tool change point, part set up, and so on.

Create and Edit a Manufacturing Program: Select the Manufacturing Program icon to add a program to the current part operation then insert all necessary program entities: machining operations, tool changes, PP instructions, and so on.

Create a Machining Process **(P2 functionality)**: Select the Machining Process icon to create a machining process, which will be stored in a CATProcess document and then as a catalog component.

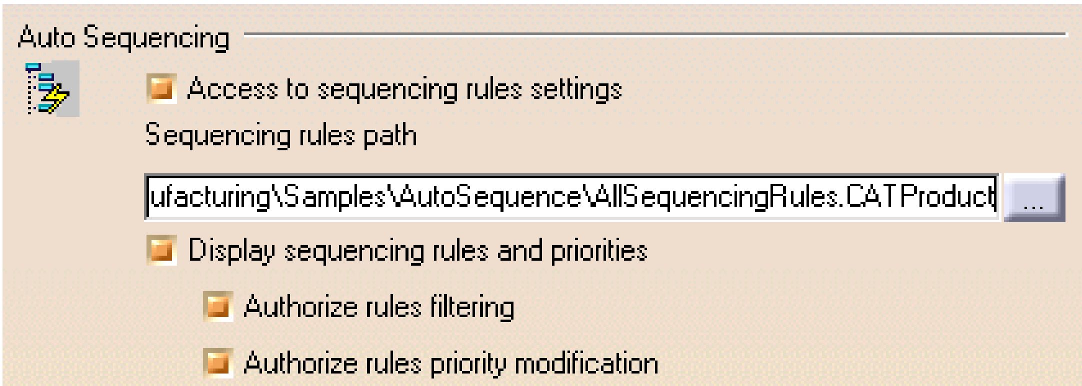
Apply a Machining Process **(P2 functionality)**: Select the Open Catalog icon to access the machining process to be applied to selected geometry.

Create a User Feature for NC Manufacturing **(P2 functionality)**.

Auto-Sequence Operations in a Program

This task shows how to optimize the order of operations in a program according to pre-defined sequencing rules.

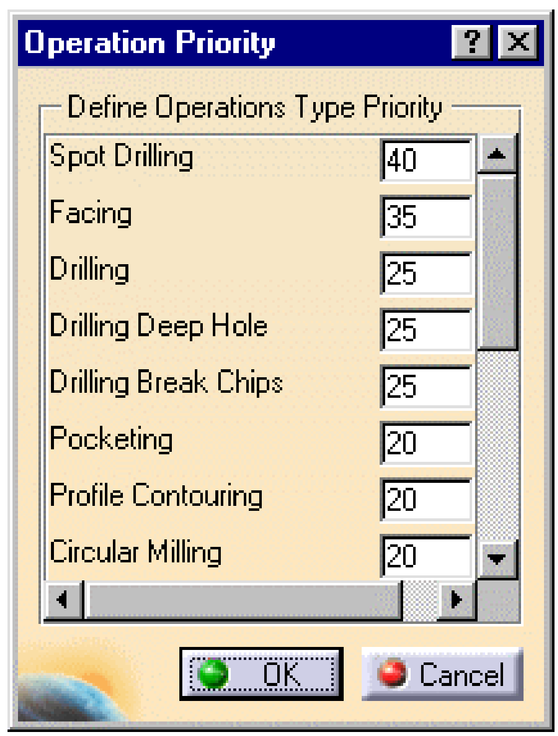
The Sequencing rules have been set up by the administrator. The Program settings under Tools > Options > NC Manufacturing are as follows:

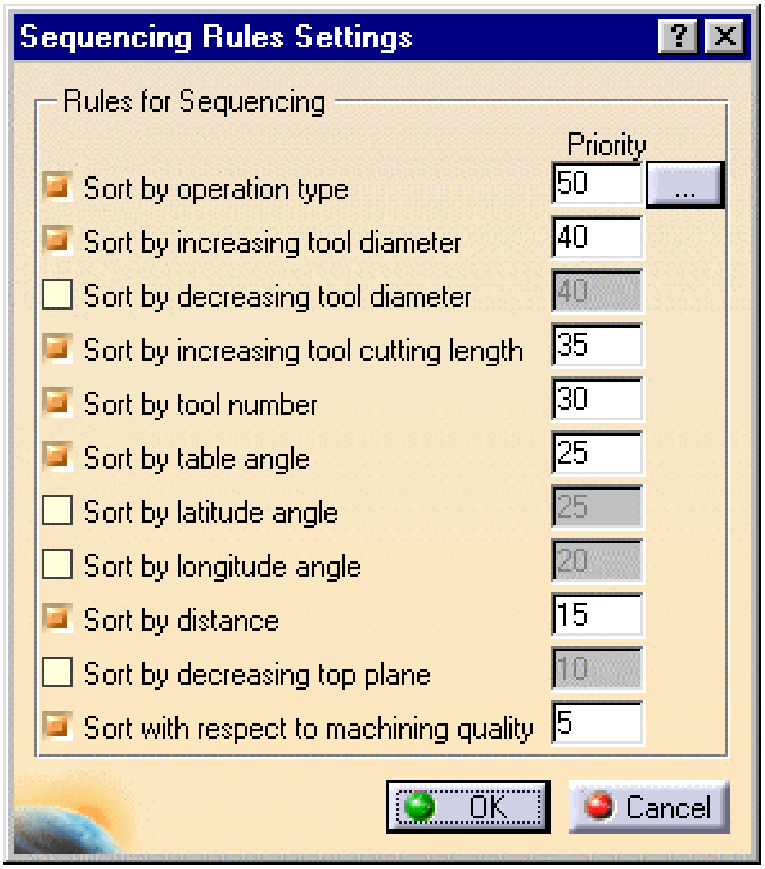


Make sure that the document in the sequencing rules path (AllSequencingRules.CATProduct in the example above) is accessible in Read/Write.

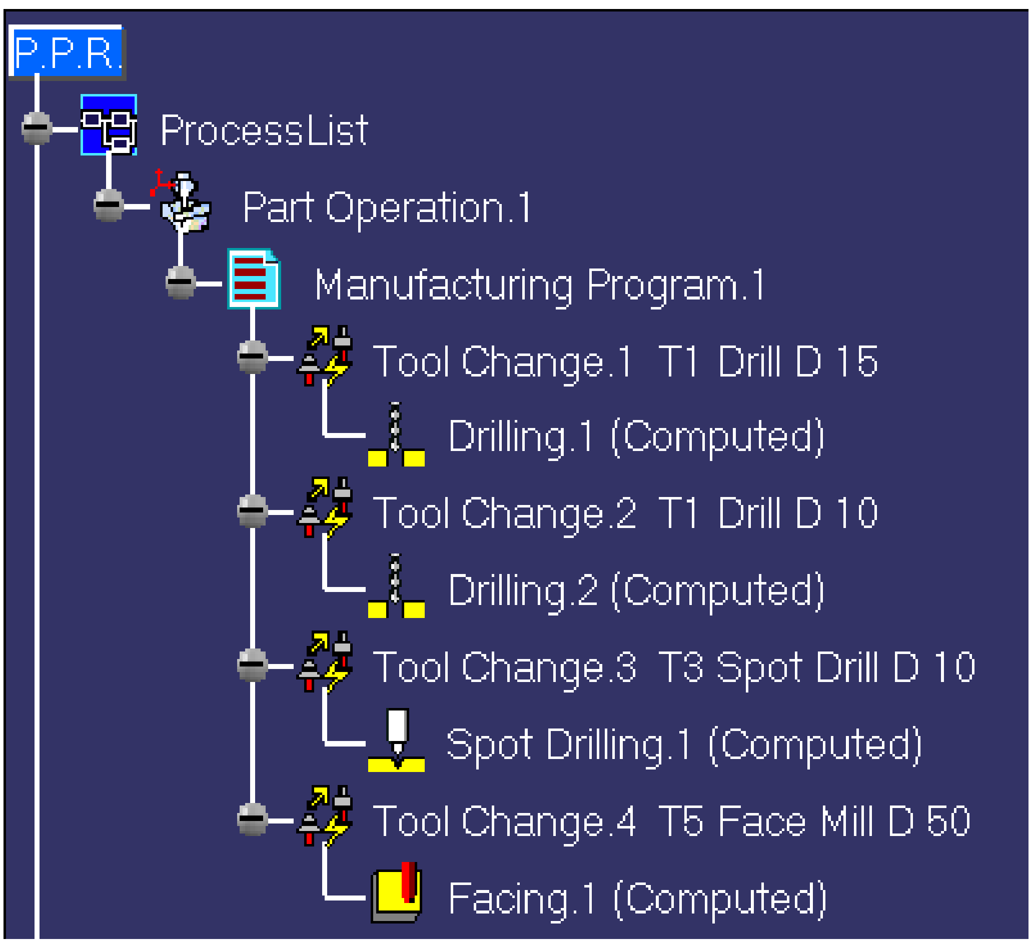
Please refer to NC Manufacturing Settings for more information.

1. Select the Rules Manager icon to visualize the administrator's sequencing rule settings.
   Then click the [...] button to visualize the sequencing priority between operations.
2.



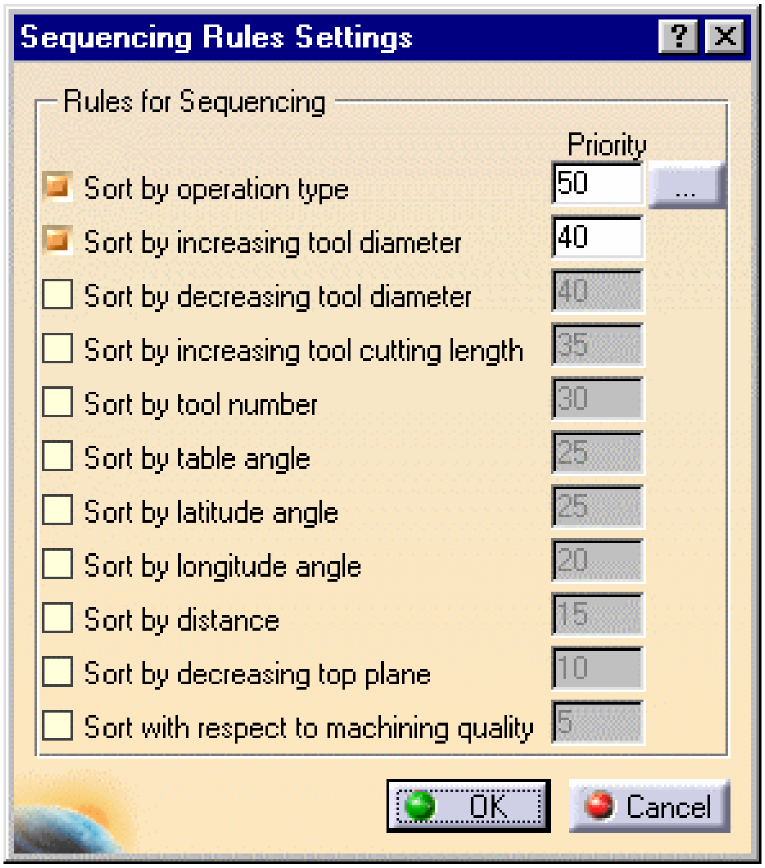


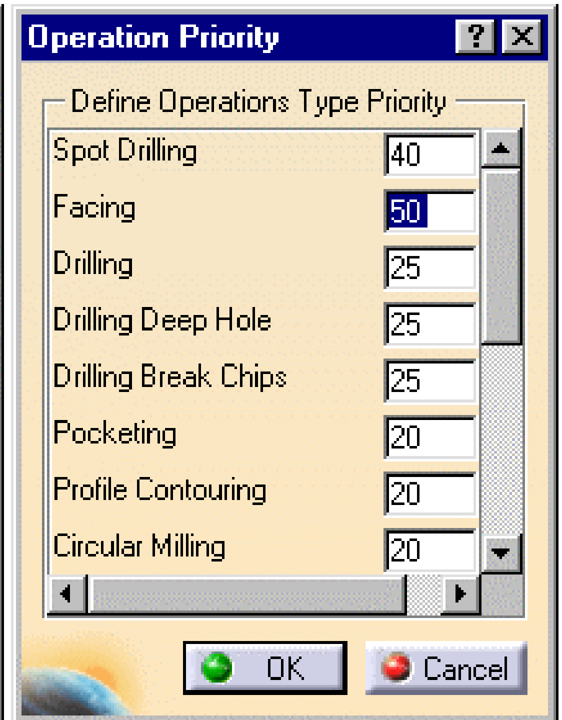
3. Create a program containing the following operations.



Select the Rules Manager icon  to change the administrator's sequencing rule settings as follows:

- de-select all rules except for Sort by operation type and Sort by increasing tool diameter
- make Facing the highest priority machining operation in the list by assigning a priority of 50. Spot drilling remains unchanged at 40 and Drilling remains unchanged at 25.

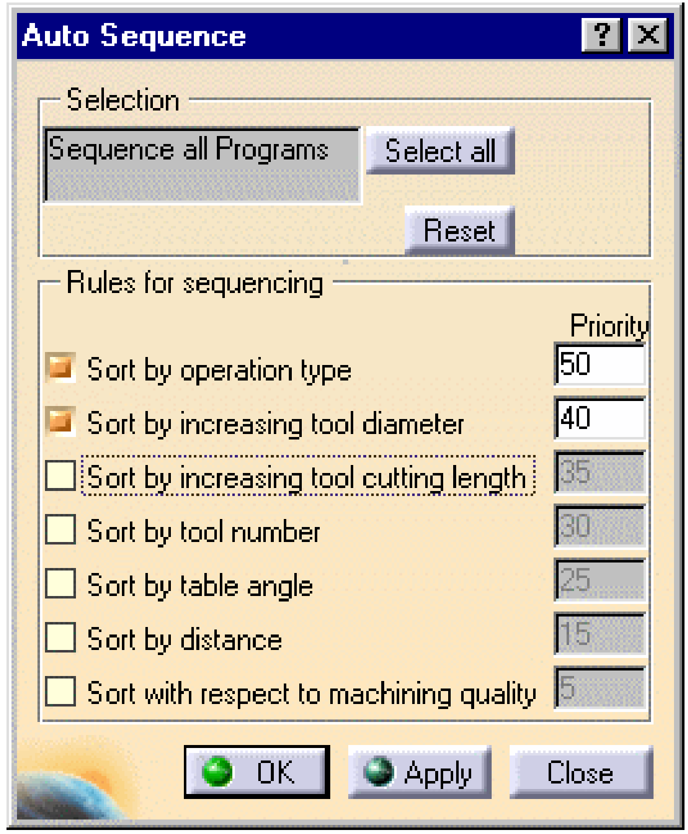




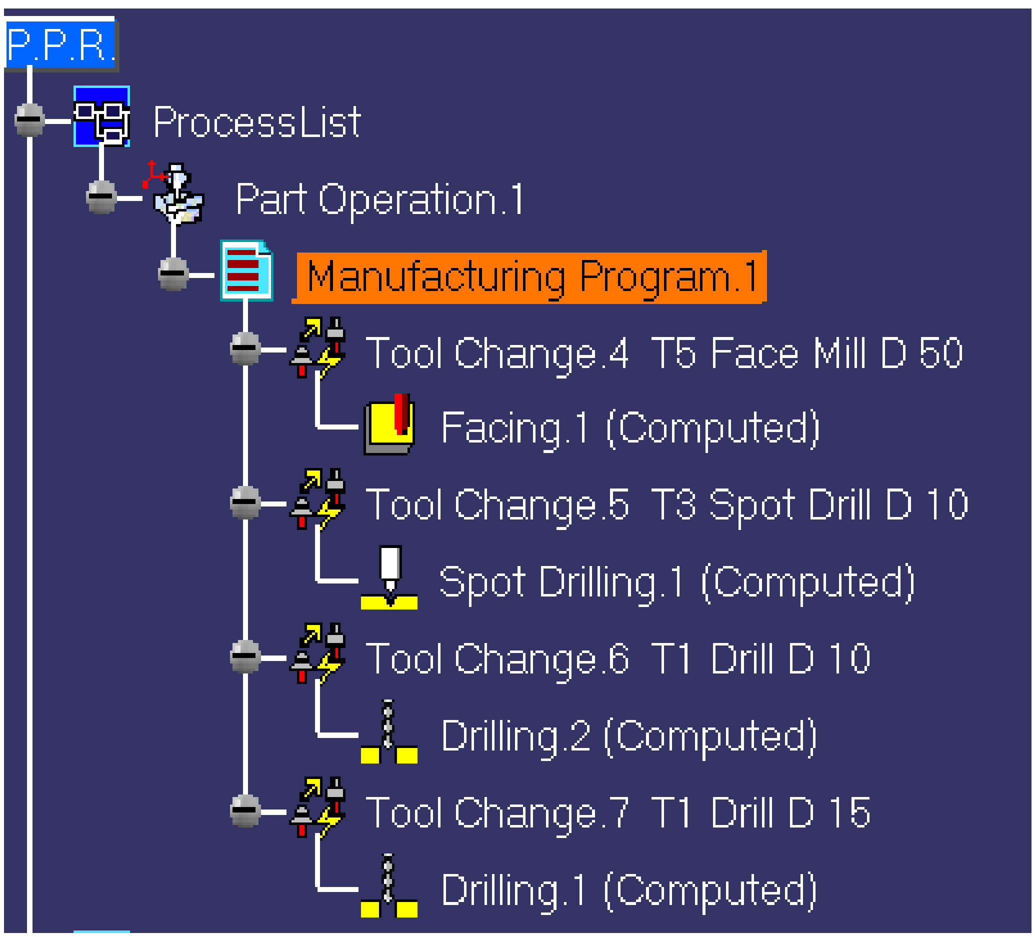
4. Select the Auto-Sequence icon  to display the Auto sequence dialog box.

   Click the **Select All** button to select all the operations of the program.

   Click **Apply** to sequence the operations according to the defined rules and priorities.



5. The program is re-sequenced as follows.



- You can click Undo if you want to cancel the auto-sequence results.
- Auto-sequencing applies to prismatic machining operations only.

Managing Manufacturing Entities

The tasks for creating and managing the specific entities of the NC manufacturing environment are documented in the *NC Manufacturing Infrastructure User's Guide*.

- Select or Create a Tool: Double click the machining operation in the program and select the Tool tab page to edit the tool characteristics or search for a new tool.
- Edit a Tool Referenced in the Program: Double click a tool referenced in the program or resource list and edit the tool characteristics in the Tool Definition dialog box.
- Specify Tool Compensation Information: Double click a tool referenced in the program or resource list and specify the tool compensation information in the Compensation tab page of the Tool Definition dialog box .
- Create and Use Machining Patterns: Select Insert > Machining Feature > Machining Pattern then select a pattern of holes to be machined.
- Feature Based Programming: Select a feature using the Manufacturing view and create operations based on this feature.
- Define Macros on a Milling Operation: Select the Macros tab page when creating or editing a milling operation, then specify the transition paths of the macros to be used in the operation.
- Define Macros on an Axial Machining Operation: Select the Macros tab page when creating or editing an axial machining operation, then specify the transition paths of the macros to be used in the operation.
- Manage the Status of Manufacturing Entities: Use the status lights to know whether or not your operation is correctly defined.

Verification, Simulation and Program Output

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

Replay Tool Path: Select the Tool Path Replay icon then specify the display options for an animated tool path display of the manufacturing program or machining operation.

Simulate Material Removal **(P2 functionality)**: Select the desired icon in the Tool Path Replay dialog box to run a material removal simulation in Photo or Video mode.

Generate APT Source Code in Batch Mode: Select the Generate NC Code in Batch Mode icon then select the manufacturing program to be processed and define the APT source processing options.

Generate NC Code in Batch Mode: Select the Generate NC Code in Batch Mode icon then select the manufacturing program to be processed and define the NC code processing options.

Generate Clfile Code in Batch Mode: Select the Generate NC Code in Batch Mode icon then select the manufacturing program to be processed and define the Clfile processing options.

Generate a CGR File in Batch Mode **(P2 functionality)**: Select the Generate NC Code in Batch Mode icon then select the manufacturing program to be processed and define the CGR file processing options.

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

Generate APT Source Code in Interactive Mode: Select the Generate NC Code Interactively icon to generate APT source code for the current manufacturing program.

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

Import an APT Source into the Program: Select the APT Import contextual command to insert an existing APT source into the current manufacturing program.

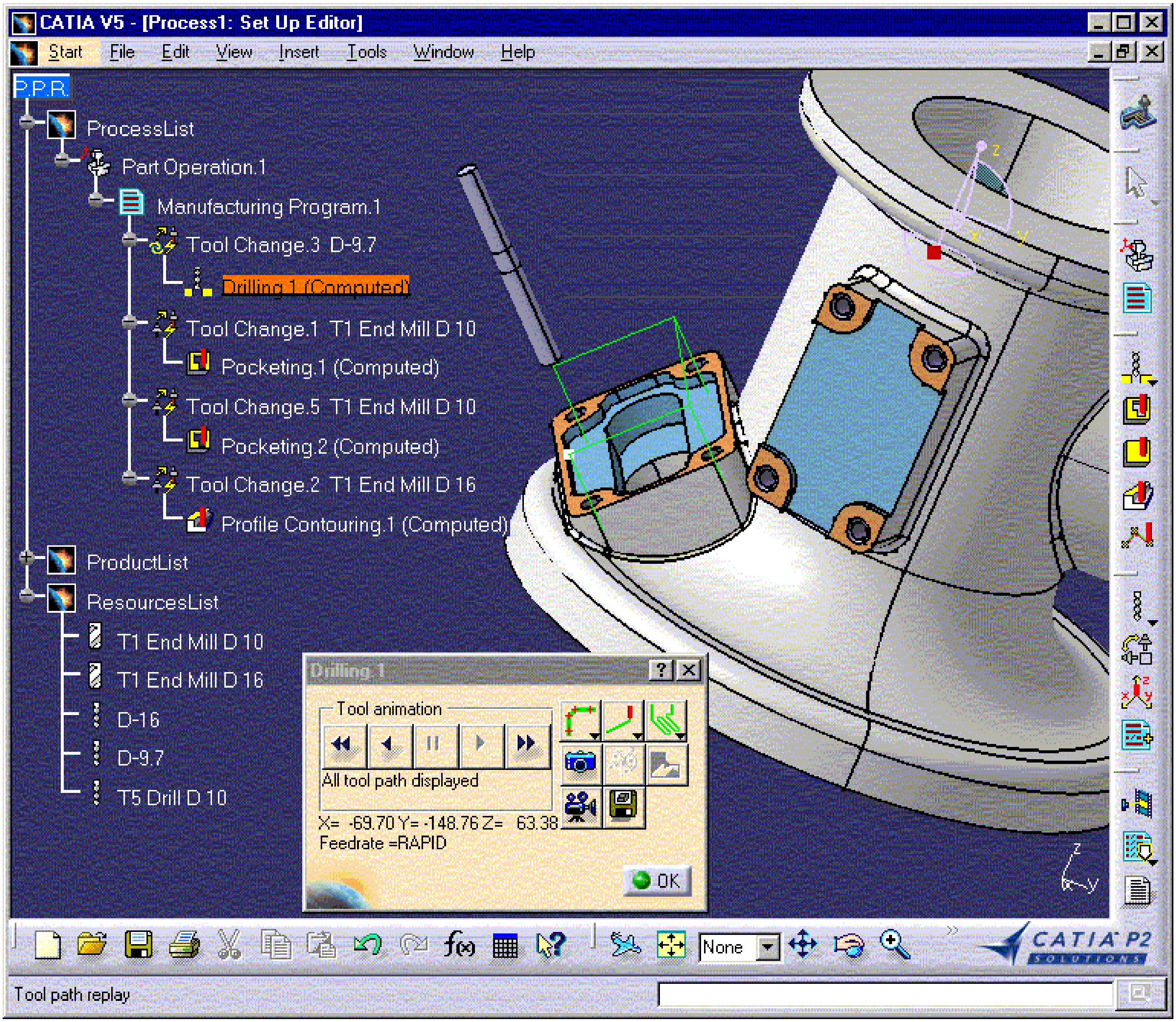
Advanced Tasks

The tasks dealing with specific NC Manufacturing processes are documented in the *NC Manufacturing Infrastructure User's Guide*.

Workbench Description

This section contains the description of the menu commands and icon toolbars that are specific to the Prismatic Machining workbench, which is shown below.

Menu Bar
Toolbars
Specification Tree



Prismatic Machining Menu Bar

The various menus and menu commands that are specific to Prismatic Machining are described below.

Start	File	Edit	View	Insert	Tools	Windows	Help

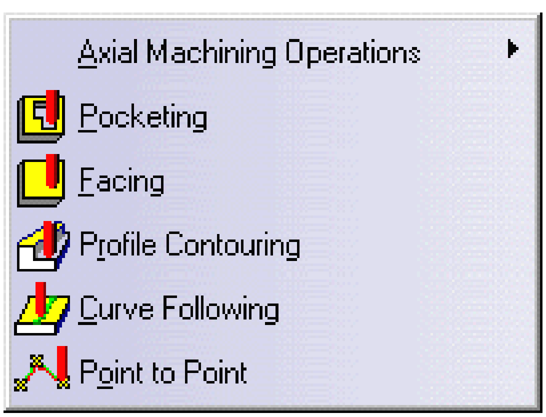
Tasks corresponding to general menu commands are described in the *CATIA Version 5 Infrastructure User's Guide*.

Tasks corresponding to common NC Manufacturing menu commands are described in the *NC Manufacturing Infrastructure User's Guide*.

Insert Menu

Command...	Description...
Prismatic Operations	See Insert > Prismatic Operations
Auxiliary Operations	Inserts Auxiliary Operations
Machining Features	Inserts Machining Features: • Prismatic Rework Area • Machining Pattern • Machining Axis System.

Insert > Prismatic Operations



Command...	Description...
Axial Machining Operations	Creates Axial Machining Operations
Pocketing	Creates a Pocketing Operation
Facing	Creates a Facing Operation
Profile Contouring	Creates a Profile Contouring Operation
Curve Following	Creates a Curve Following Operation
Point to Point	Creates a Point to Point Operation

Prismatic Machining Toolbars

The Prismatic Machining workbench includes specific icon toolbars:

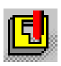
- Prismatic Operations
- Machining Features
- Manufacturing Program Optimization.

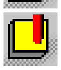
The other toolbars in the workbench are common to all the NC Manufacturing products and are described in the *NC Manufacturing Infrastructure User's Guide*.

Prismatic Operations Toolbar

This toolbar contains the commands for creating and editing 2.5 axis Milling and Axial Machining operations.

The icons for creating and editing 2.5 axis Milling operations are as follows.

 See Create a Pocketing Operation

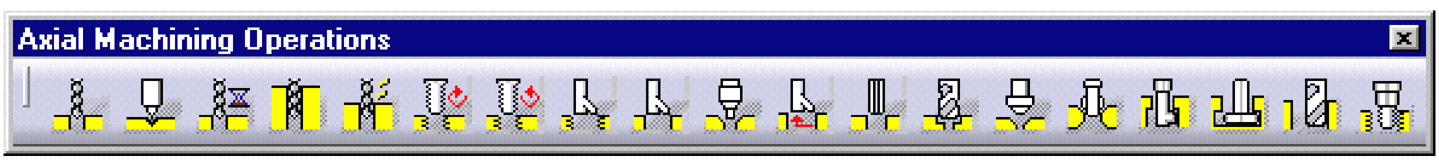
 See Create a Facing Operation

 See Create a Profile Contouring Operation

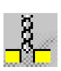
 See Create a Curve Following Operation

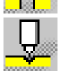
 See Create a Point to Point Operation

The following toolbar is accessed from the drop-down icon in the Prismatic Operations toolbar.

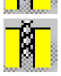


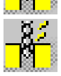
It contains icons for creating and editing Axial Machining operations as follows.

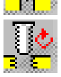
 See Create a Drilling Operation

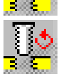
 See Create a Spot Drilling Operation

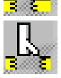
 See Create a Drilling Dwell Delay Operation

 See Create a Drilling Deep Hole Operation

 See Create a Drilling Break Chips Operation

 See Create a Tapping Operation

 See Create a Reverse Threading Operation

 See Create a Thread without Tap Head Operation

See Create a Boring Operation

See Create a Boring and Chamfering Operation

See Create a Boring Spindle Stop Operation

See Create a Reaming Operation

See Create a Counterboring Operation

See Create a Countersinking Operation

See Create a Chamfering Two Sides Operation

See Create a Back Boring Operation

See Create a T-Slotting Operation

See Create a Circular Milling Operation

See Create a Thread Milling Operation

Machining Features Toolbar

This toolbar contains the command for creating a **prismatic rework area** feature.

See Create Operations for Channel and Corner Rework for information about how to use this feature

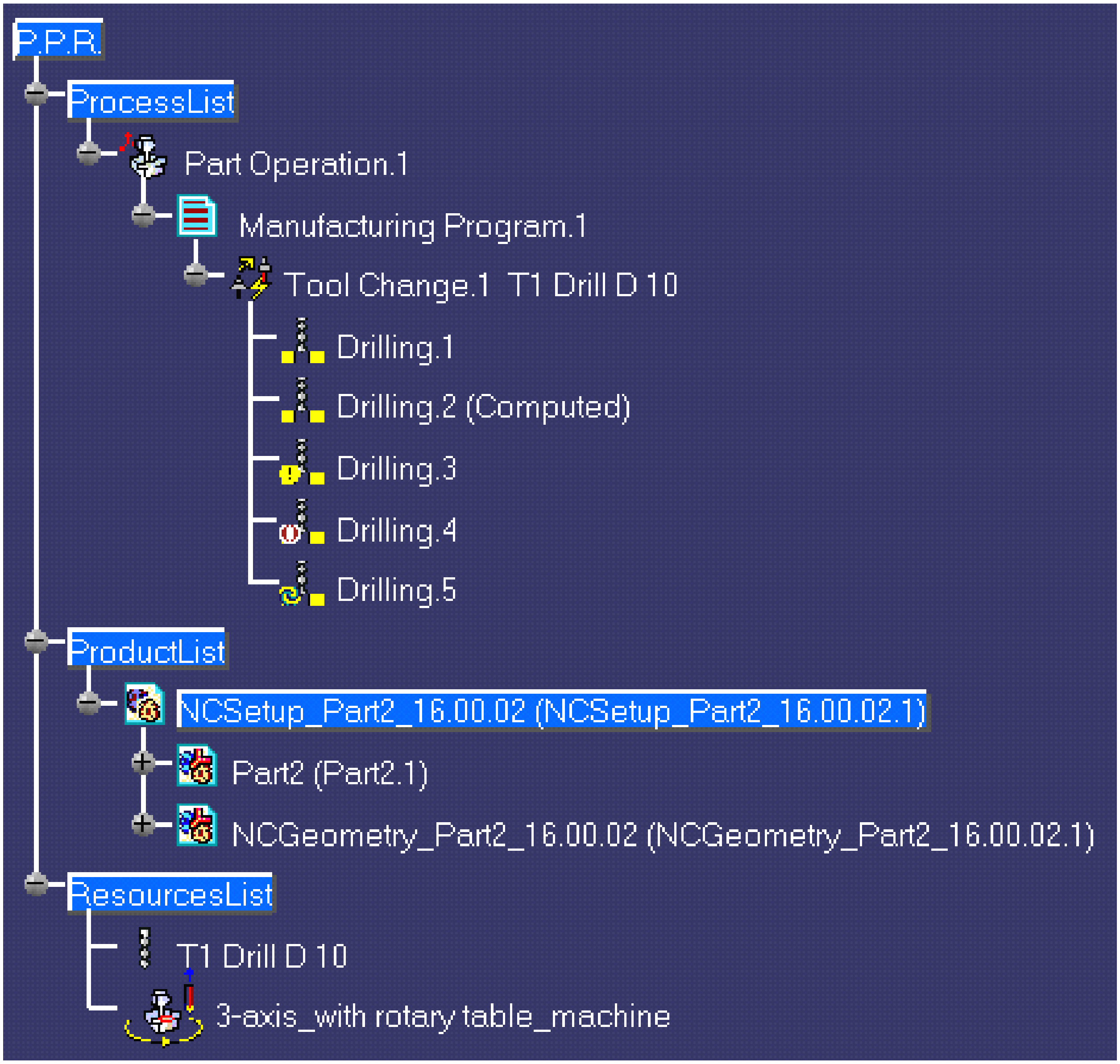
Manufacturing Program Optimization Toolbar

This toolbar contains the commands for optimizing the order of operations in the program according to pre-defined sequencing rules.

See Auto-sequence operations of a program according to pre-defined rules

Specification Tree

Here is an example of a Process Product Resources (PPR) specification tree for Prismatic Machining.



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

- **Part Operation** defines the manufacturing resources and the reference data.
- **Manufacturing Program** is the list of all of the machining operations, associated tool changes, and auxiliary operations. The example above shows that:
    - Drilling.1 is complete and has not been computed
    - Drilling.2 is complete but has been computed (by means of a replay)
    - Drilling.3 does not have all of the necessary data (indicated by the exclamation mark symbol)
    - Drilling.4 has been deactivated by the user (indicated by the brackets symbol)
    - Drilling.5 has been modified and needs to be recomputed (indicated by the update symbol).

Product List gives all of the parts to machine as well as CATPart documents containing complementary geometry.

Resources List gives all of the resources such as machine or tools that can be used in the program.

Customizing

The tasks for customizing your NC Manufacturing environment are documented in the *NC Manufacturing Infrastructure User's Guide*.

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

Reference Information

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

Glossary

A

approach macro	Motion defined for approaching the operation start point
auxiliary command	A control function such as tool change or machine table rotation. These commands may be interpreted by a specific post-processor.
axial machining operation	Operation in which machining is done along a single axis and is mainly intended for hole making (drilling, counter boring, and so on).

B

back and forth	Machining in which motion is done alternately in one direction then the other. Compare with one way.
bottom plane	A planar geometric element that represents the bottom surface of an area to machine. It is normal to the tool axis.

C

clearance macro	Motion that involves retracting to a safety plane, a linear trajectory in that plane and then plunging from that plane.
climb milling	Milling in which the advancing tool rotates down into the material. Chips of cut material tend to be thrown behind the tool, which results to give good surface finish. Compare with conventional milling.
conventional milling	Milling in which the advancing tool rotates up into the material. Chips of cut material tend to be carried around with the tool, which often impairs good surface finish. Compare with climb milling.

D

DPM	Digital Process for Manufacturing.

E

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

F

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

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

feedrate Rate at which a cutter advances into a work piece.
Measured in linear or angular units (mm/min or mm/rev, for example).

fixture Elements used to secure or support the workpiece on a machine.

G

gouge Area where the tool has removed too much material from the workpiece.

H

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

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

I

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

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

L

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

M

machine rotation An auxiliary command in the program that corresponds to a rotation of the machine table.

machining axis system Reference axis system in which coordinates of points of the tool path are given.

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

machining operation Contains all the necessary information for machining a part of the workpiece using a single tool.

machining tolerance The maximum allowed difference between the theoretical and computed tool path.

manufacturing process Defines the sequence of part operations necessary for the complete manufacture of a part.

manufacturing program Describes the processing order of the NC entities that are taken into account for tool path computation: machining operations, auxiliary commands and PP instructions.

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

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

O

offset Specifies a virtual displacement of a reference geometric element in an operation (such as the offset on the bottom plane of a pocket, for example). Compare with thickness.

one way Machining in which motion is always done in the same direction. Compare with zig zag or back and forth.

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

P

part operation Links all the operations necessary for machining a part based on a unique part registration on a machine. The part operation links these operations with the associated fixture and set-up entities.

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

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

Point to Point operation A milling operation consisting of a sequence of elementary tool motions between points.
A tool motion can be defined by:

- the point the tool tip has to reach (Goto Point motion)
- positioning the tool in contact with a part element, a drive element and possibly a check element, while taking To/On/Past conditions into account (Goto Position motion).

PP instruction Instructions that control certain functions that are auxiliary to the tool-part relationship. They may be interpreted by a specific post processor.

PPR Process Product Resources.

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

R

retract macro Motion defined for retracting from the operation end point

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

S

safety plane A plane normal to the tool axis in which the tool tip can move or remain a clearance distance away from the workpiece, fixture or machine.

set up Describes how the part, stock and fixture are positioned on the machine.

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

spindle speed The angular speed of the machine spindle.
Measured in linear or angular units (m/min or rev/min, for example).

stock Workpiece prior to machining by the operations of a part operation.

T

thickness Specifies a thickness of material to be removed by machining.
Compare with offset.

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

tool axis Center line of the cutter.

tool change An auxiliary command in the program that corresponds to a change of tool.

tool clash Area where the tool collided with the workpiece during a rapid move.

tool path The path that the center of the tool tip follows during a machining operation.

total depth The total depth including breakthrough distance that is machined in a hole making operation.

U

undercut Area where the tool has left material behind on the workpiece.

Z

zig zag Machining in which motion is done alternately in one direction then the other. Compare with one way.

Index

E

F

G

H

I

J

M

N

O

P

T

U

Z