

NC Tools	10
Facing Tool	10
End Mills	11
Center Drills	12
Spot Drills	13
Drill	14
Countersink	15
Counterbore Mill	16
Reamer	17
Boring Bar	18
Tap	19
T-Slotter	20
Multi-Diameter Drill	21
Two Sides Chamfering Tool	22
Boring and Chamfering Tool	23
Conical Mill	24
Thread Mill	25
Milling Directions	26
Conventional Milling	26
Climb Milling	27
Prismatic Machining Workbench	28
Specifications Tree	28
Toolbars	30
Preparing to Machine	33
Part Design Review	33
Measurement Review	37
Assembly Review	40
Part Operation Setup	45
Defining the Part Operation	45
Machine Definition	46
Machining Axis Definition	50
Geometry Definition	52
Replaying	57
Photo Mode	67
Video Mode	77

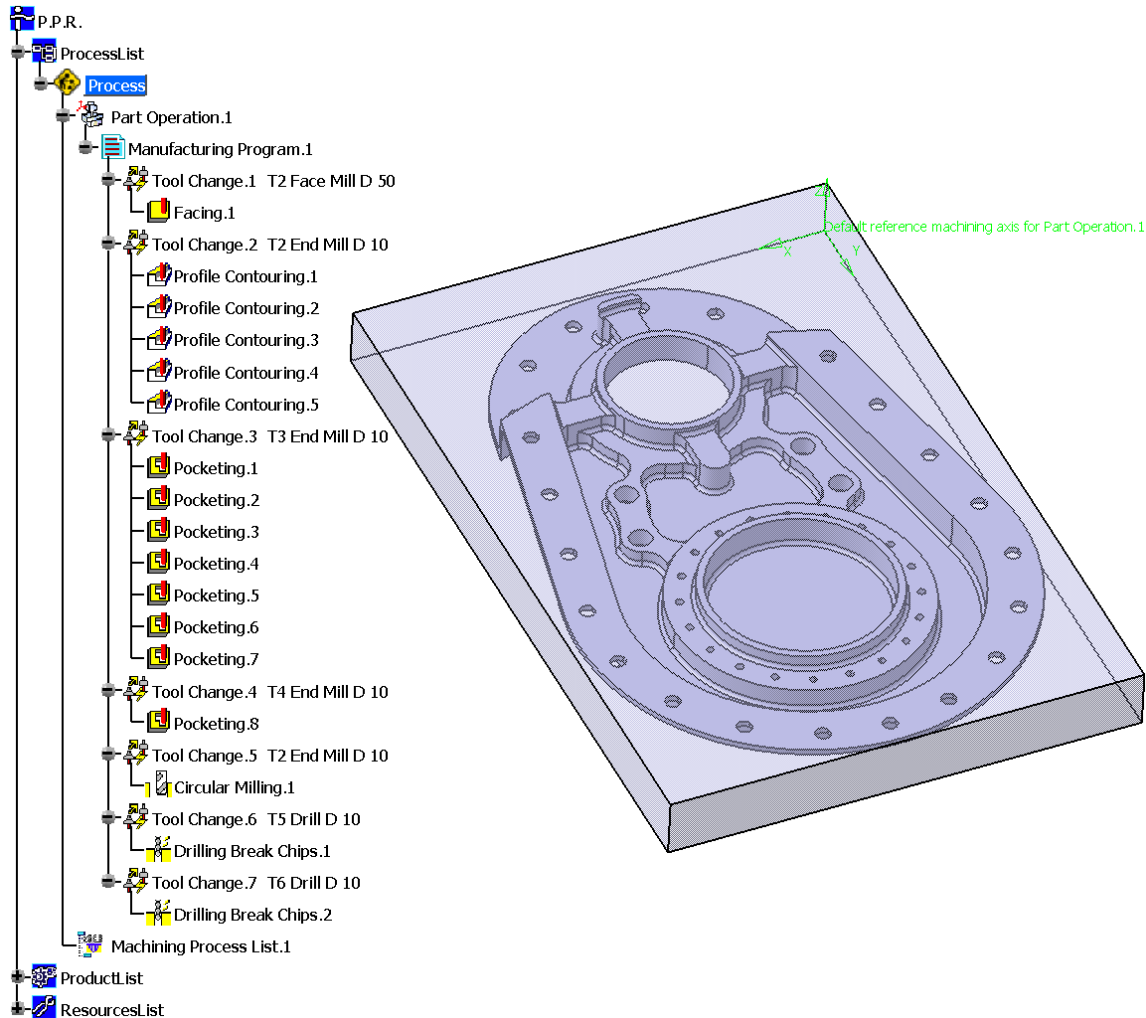
Machining	85
Facing	86
Facing Geometry tab	88
Edge Selection	92
By Belt of Faces	92
By Boundary of Faces	93
Sectioning	93
Tool tab	95
Facing Machining Parameters tab	98
Speeds and Feeds tab	117
Macros tab	120
Profile Contour Milling	137
Profile Contouring Geometry Tab	138
Profile Contouring Modes	141
Profile Contouring Machining Parameters Tab	152
Profile Contouring Feeds and Speeds Tab	161
Groove Milling	181
Groove Milling Machining Parameters Tab	182
Pocketing	189
Pocketing Machining Parameters Tab	196
Manually Defining Tool Changes	205
Prismatic Roughing	235
Curve Following	249
Point to Point	257
Axial Machining	265
Spot Drilling	267
Geometry Tab	268
Drilling	278
Drilling Dwell Delay	281
Drilling Deep Hole	281
Drilling Break Chips	282
Tapping & Reverse Tapping	282
Threat Without Tap Head	283
Boring	283
Boring and Chamfering	283
Boring Spindle Stop	284
Reaming	284
Counter Boring	284
Counter Sinking	285
Chamfering 2 Sides	285
Back Boring	286
T-Slotting	286
Circular Milling	287
Thread Milling	289

Patterns	290
Part Design Patterns	290
Machining Patterns	291
 Multiple Part Operations	 299
 Fixtures	 309
Clamps	309
Tabs	320
 Transformations	 337
Copy-Transformation	338
COPY Operator Instruction	347
TRACUT Operator Instruction	359
Opposite Hand Machining	363
 NC Documentation	 369
 APT Code Generation	 377
 Practice Problems	 383
 Appendix A	 397
NC Manufacturing - General	397
NC Manufacturing - Resources	398
NC Manufacturing - Operation	399
NC Manufacturing - Output	400
NC Manufacturing - Program	401
NC Manufacturing - Photo Video	402

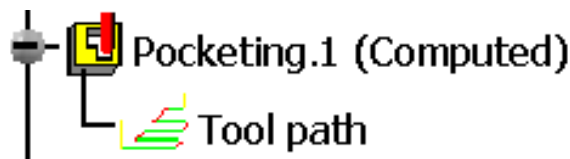
Replaying

Before you can fully understand what the various machining options allow you to do, you must first be familiar with replaying your tool paths. Replaying, as you might guess, is the most important part of verifying whether the tool is accurately cutting the part, as well as viewing to insure the correctness of the program overall.

Open A Replay from the *Replay* folder. This machining process already had machining operations applied to it. The various machining operations will allow you to get familiar with replaying your machining processes.

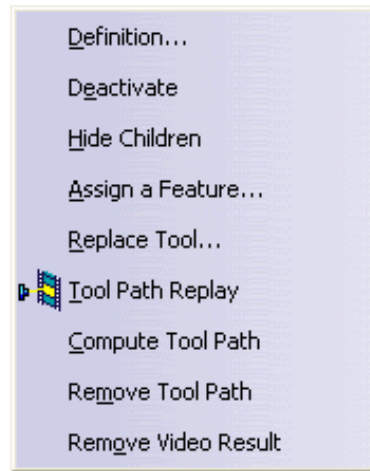


Machining operations can have two states to them. One state is to have the tool paths computed. In this state, the tool paths are available for the machining operation, and can be replayed instantly. When the tool paths are computed, a (*Computed*) text will appear next to the operation, and an additional branch will show up below the machining operation.



Since none of the machining operations have this extra branch and the *(Computed)* text, the tool paths are not present. The first step before replaying will be to compute the tool paths. Computing the tool paths can be done a number of different ways.

With the third mouse button, select on *Facing.1* from the tree. Select *Facing.1* object from the bottom of the contextual list. This will display the object properties for the facing operation. The last four options are ones that are most important to us while replaying.



Tool Path Replay

This will replay the currently selected machining operation. This can also be accomplished by selecting the replay icon in the left hand toolbar. If a tool path is not computed, replaying the tool path will automatically compute it.

Compute Tool Path

This will allow you to manually compute the tool path. When you compute the tool path, you also have the option to force the tool path computation.

Remove Tool Path

This allows you to remove or strip the tool path out of the process. By removing the tool path, you will dramatically reduce the size of the file saved.

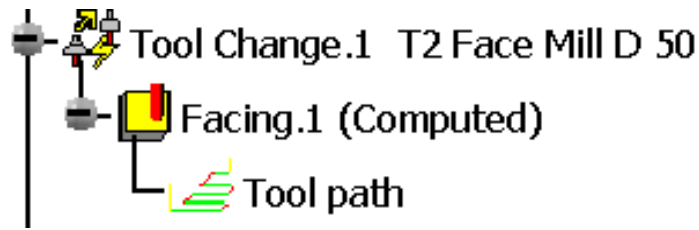
Remove Video Result

When a video result is created, a temporary image is stored in your profile. This temporary image is generally deleted when you log off of your computer, hence breaking the link with CATIA. Generally it is a good idea to remove the video result when saving the file.

Select *Compute Tool Path*. This will display the *Computation* window that asks if you want to compute if necessary or force the computation.

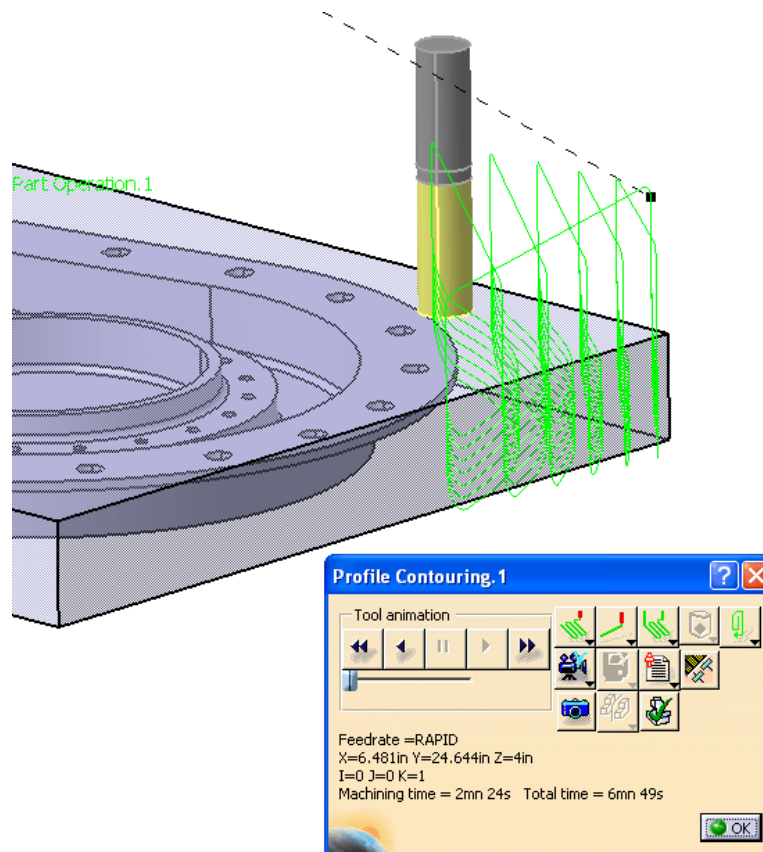
Select *Computation if necessary*, then select *OK*. Since the tool paths were not computed, they will automatically compute. An information window displays the number of tool paths computed.

Select *OK* to the *Manufacturing Information* window. Now the tool path has been computed.



You can also compute the tool paths by replaying the operation.

With the third mouse button, select on *Profile Contouring.1* from the tree. Select the *Profile Contouring.1* object from the bottom of the contextual list. Select *Tool Path Replay*. The *Tool Path Computation* window will show while the tool path is computed. This will display the tool paths on the screen, as well as show the *Replay* window.



Select *OK* to the window.


Computing the tool paths one at a time can become rather trying, especially when you have a lot of machining operations. Fortunately, you can also compute the tool paths by Manufacturing Program.

With the third mouse button, select on *Manufacturing Program.1* from the tree, then select on the *Manufacturing Program.1* object. Here there will be a number of new options, however, you will still find the ability to compute, remove, and replay the tool paths.

Select **Compute Tool Path**. This will display the *Computation* window again.

Select **OK**. Since the option for *Computation if necessary* is checked by default, you do not need to re-select it. The first two tool paths have already been computed, hence they will not get re-computed in this case. If you had selected on *Forced computation*, the first two tool paths would get re-computed.

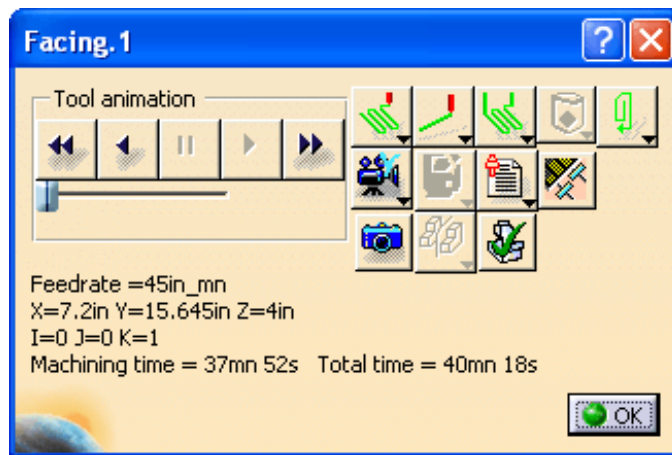
This will now go through and compute the remainder of the tool paths. For long programs this may take some time. Once done, the *Manufacturing Information* window will display indicating the number of tool paths computed.

With no machining operations selected, select the tool path replay icon from the tool bar, then select **Pocketing.8**.  Notice the tool paths immediately show up. Since the tool paths were already computed, there is no delay for tool path computation.

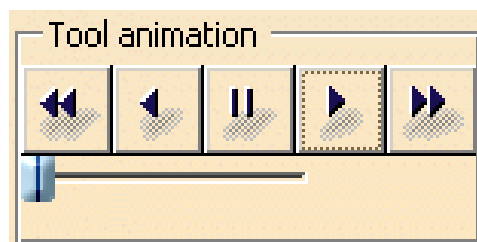
Select **OK** to the window.

Now it is time to investigate the replay window a bit closer.

Select the tool path replay icon, then select the *Facing.1* machining operation. The replay window is shown.




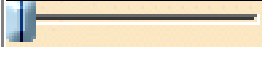


Tool Animation This frame houses the player controls. The controls work similar to VCR controls.



 – F5 – *Skip to Beginning* Skips the tool to be beginning of the tool path

 – F6 – *Play Backwards* Replays the tool paths backwards

-  – *Pause* Pauses the tool path replay
-  – F7 – *Play Forwards* Plays the tool path forwards
-  – F8 – *Skip to End* Jumps the tool path to **the end**
-  – *Speed* Allows you to speed up or slow down the tool path animation replay. *Note: In Prismatic Machining, many of the tool paths run fast, and cannot be slowed down. The speed control does come in useful when working with Surface Machining operations.*

```
Feedrate =1143mm_mn
X=7.2in Y=15.645in Z=4in
I=0 J=0 K=1
Machining time = 37mn 52s Total time = 40mn 53s
```

Information block This area of the window gives information about the various aspects of the tool and the replay motion.


Feedrate Indicates the feedrate at any given point for the motion

X= Y= Z= Displays the current X, Y, and Z location of the tool


I= J= K= Denotes the I, J, and K components of the tool axis


Machining Time Indicates the time that the tool is cutting

Total Time This is the overall time of the process. This time is an addition of the machining time, as well as any air time that the tool makes.

 *Replay Mode* Controls how the tool paths are replayed. There are five different choices for the replay mode.



 *Continuous* The tool will replay from the current position and will not stop until it reaches **the end** of the operation

 *Plane by plane* The tool stops each time it moves to a different axial location. This option is best suited to allow you to visually inspect each axial pass of the tool.

以上内容仅为本文档的试下载部分，为可阅读页数的一半内容。如要下载或阅读全文，请访问：<https://d.book118.com/426105220003010213>