PartMaker/SwissCAM 2015

Getting Started

Metric version

Important User Notices

Delcam Ltd has no control over the use made of the software described in this manual and cannot accept responsibility for any loss or damage howsoever caused as a result of using the software. Users are advised that all the results from the software should be checked by a competent person, in accordance with good quality control procedures.

The functionality and user interface in this manual is subject to change without notice in future revisions of the software.

The software described in this manual is furnished under licence agreement and may be used or copied solely in accordance with the terms of such licence.

Delcam Ltd grants permission for licensed users to print copies of this manual or portions of this manual for personal use only. Schools, colleges and universities that are licensed to use the software may make copies of this manual or portions of this manual for students currently registered for classes where the software is used.

Acknowledgements

This documentation references a number of registered trademarks and these are the property of their respective owners. For example, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States.

Patents

PartMaker software is subject to the following patents:

Patent granted: US 6, 112, 133 Visual system and method for generating a CNC program for machining parts with planar and curvilinear surfaces

Patent granted: US 6, 741, 905 Visual system for programming of simultaneous and synchronous machining operations on lathes

The Raceline smoothing functionality is subject to patent applications.

Patent granted: GB 2374562 Improvements Relating to Machine Tools

Patent granted: US 6,832,876 Machine Tools

The Vortex machining functionality is subject to patent applications.

Patent application: 1121277.6 Adaptive Clearance

PartMaker 2015. Published on 27 November 2014

Contents

About this guide

This **Getting Started** guide provides step-by-step instructions to guide you through the process of programming a part in PartMaker/SwissCAM.

This guide assumes that you know how to operate your machine and how to select suitable tools and cutting conditions. ***If you are unsure about any aspect of operating your machine, consult an expert or seek advice from your machine supplier.***

The machining parameters used in the examples in this guide have been selected to illustrate the effects of various settings in PartMaker. ***These values may not be suitable for cutting on your particular CNC machine. If you want to machine any parts based on the examples given, carefully review and adjust the parameters to ensure safe cutting conditions.***

To find out more about PartMaker/SwissCAM, click  on PartMaker's **Main** toolbar to display the PartMaker online help.

Introduction to PartMaker

PartMaker/SwissCAM automates the tedious task of manually programming parts for sliding headstock, Swiss-type lathes. The key stages involved in programming a part in PartMaker are:

Programming the part

PartMaker's patented visual approach to programming enables you set up machining functions, such as turning, plane milling, and cylinder milling, in separate 2D planes (known as Face windows). In this way, you can break a part down into a series of much simpler operations.

You can create your part geometry within PartMaker or you can import CAD files that have been created in other CAD programs.

Creating a Process Table

When you have programmed the toolpaths for a part, you can generate a Process Table to view the processes required to machine the part you have programmed. By modifying the Process Table, you can synchronize processes being performed on separate spindles or by different tool posts, thus maximizing the efficiency and productivity of your CNC machine.

Simulating toolpaths

Once you have generated a Process Table, you can view a 3D simulation of the processes shown in the Process Table. This enables you to identify any errors before the part is machined.

Creating an NC Program

When you are happy with the toolpaths and the simulation, the next step is to generate an NC Program for a specific machine. This process is often referred to as post processing, because PartMaker uses a post configuration (.pst) file to convert the information in the Process Table into an NC Program. The Post Configuration files available to you depend on your PartMaker/SwissCAM license.

About PartMaker files

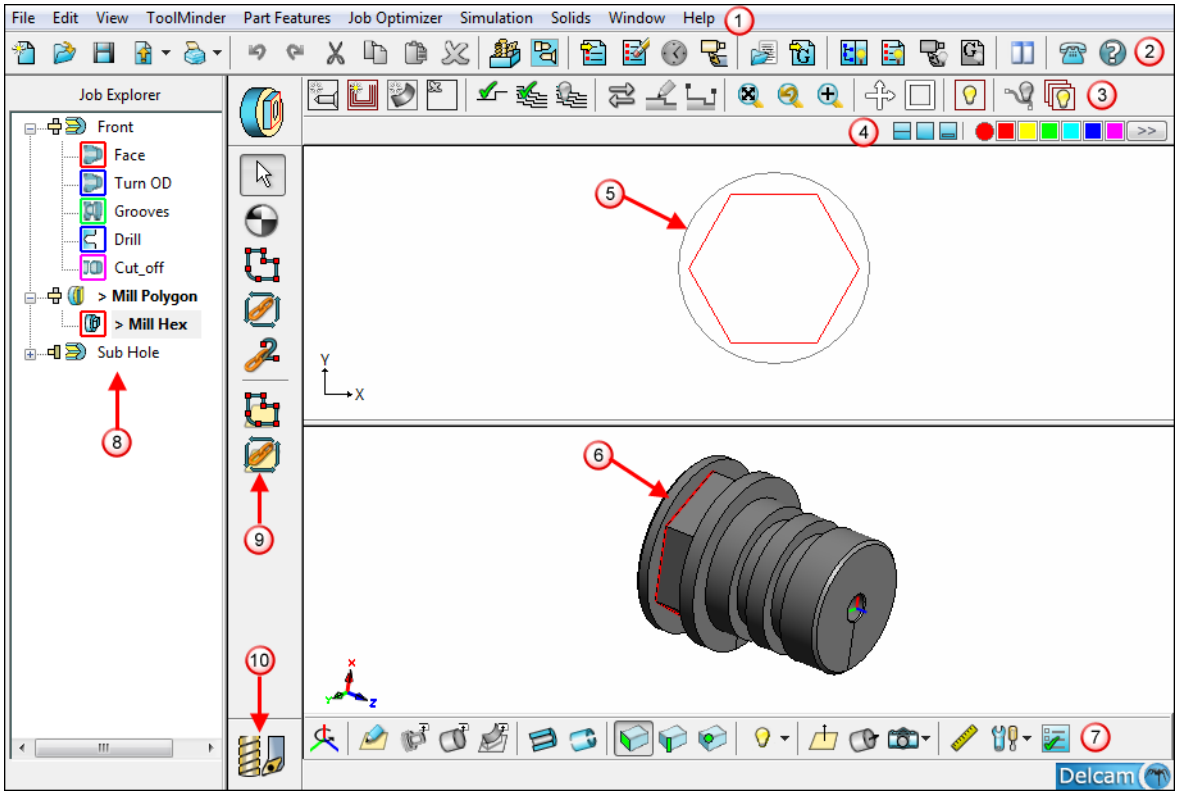
When working in PartMaker, you use the following types of files:

- Tools database (.tdb) files — These store information about tools in the tool crib and so enable you to keep track of your current tool inventory.
- Cycles database (.cdb) files — In PartMaker, you can combine several repetitive operations (such as, center drilling, drilling, tapping, and boring) into a single entity called a cycle. You can also allocate a specific tool from the Tools database to a cycle.
- Materials database (.mdb) files — These store information about the materials you use. PartMaker uses the information from the Materials database to automatically calculate feed rates and spindle speeds.
- Job (.job) files — These store details of the toolpaths you create to machine the part. Each job file references database files for tools, cycles and materials, so it is easy to reuse tool, cycle, and material information in many different job files.
- Postprocessor configuration (.pst) files — These store the information required to convert the details shown in PartMaker's Process Table into an NC Program file that is suitable for a particular machine.

PartMaker also enables you to:

- Import information from files created in other programs. For example, importing 3D solid models created in other CAM programs.
- Export information from PartMaker to files that can be used by other programs. For example, exporting geometry created in PartMaker to .dxf files suitable for use with AutoCAD.

About the main PartMaker window



1 The PartMaker menu bar contains menu options for performing tasks within PartMaker.

2 Use the **Main toolbar** to open and close files and to control the display of windows within the main PartMaker window.

3 Use the **CAM window toolbar** to create new part features, verify toolpaths and modify the view within the Face window.

4 This area displays the **Color Palette** for selecting the color of profiles or geometry. It also displays buttons for controling the size of fixed Face windows

5 The 2D **Face window** displays the 2D part geometry and this is where you assign profiles to toolpaths.

6 The 3D **Solids window** displays a 3D solid model.

7 Use the **Solids Window toolbar** to work with solids models and control the view within the Solids window.

8 The **Job Explorer pane** lists the Face windows in the .job file, and the part features programmed in each Face window.

9 Use the **Profile toolbar** to create profiles for toolpaths.

10 Use **CAD/CAM Switch** button to move between CAD and CAM modes.

Using the mouse in PartMaker

Mouse button 1 (usually the left mouse button)

Use the left mouse button to perform the following actions:

Window	Action
Face window	Click to select a profile in the 2D wireframe view.
Solids window	Click to select an element (such as surfaces, curves and vertices) on the solid model. Double-click to display details of the element you have selected. Hold down this button and drag the mouse to rotate the solid model.
Simulation window	Hold down this button and drag the mouse to rotate the view displayed in this window.

Mouse button 2 (usually the center scroll wheel)

Use the center mouse button to perform the following actions:

Window	Action
Face window	Rotate this button to zoom in and out of the wireframe view. Hold down this button and drag the mouse to reposition the wireframe view.
Solids window	Rotate this button to zoom in and out of the Solids view. Hold down this button and drag the mouse to reposition the Solids view.
Simulation window	Rotate this button to zoom in and out of the Simulation view. Hold down this button and drag the mouse to reposition the Simulation view.

Mouse button 3 (usually the right mouse button)

Click the right mouse button to display a context menu, which displays menu options relevant to the window in which you are working.

Starting PartMaker/SwissCAM

1. Double-click the PartMaker/SwissCAM shortcut on your desktop:

   PartMaker opens and the **Setup** dialog is displayed.

2. Click the **Close** button to close the **Setup** dialog. You will learn how to use this dialog later in this Getting Started guide.

   A Face window, called **Front**, is displayed ready for you to work in PartMaker.

3. Before programming the part, you need to open the Tools and Cycles files you want to work with. To do this:

   a. Select **Files > Open Tools File** to display the **Open Tools File** dialog. Browse to your PartMaker User Files folder and select the getting_started_metric.tdb file in the Getting_Started_Tutorial subfolder.

   b. Select **Files > Open Cycles File** to display the **Open Cycles File** dialog. Browse to your PartMaker installation folder and select the getting_started_metric.cdb file in the Getting_Started_Tutorial subfolder.

   You will learn how to add new tools and cycles to these files later in this Getting Started guide.

Creating geometry

The geometry of a part is a key element to creating the toolpaths that define how the part will be machined.

In PartMaker, you can create geometry in any of the following ways:

- Import a 3D solid model file, as described in Importing a 3D solid model and extracting geometry (see page 7).
- Import a 2D (.dxf) file, as described in Importing a 2D file (see page 10).
- Create CAD geometry within PartMaker using its CAD mode. For details of how to create CAD geometry directly within PartMaker, refer to PartMaker's online help.

The ability to import a 3D solid model into PartMaker is available as a cost option.

Importing a 3D solid model and extracting geometry

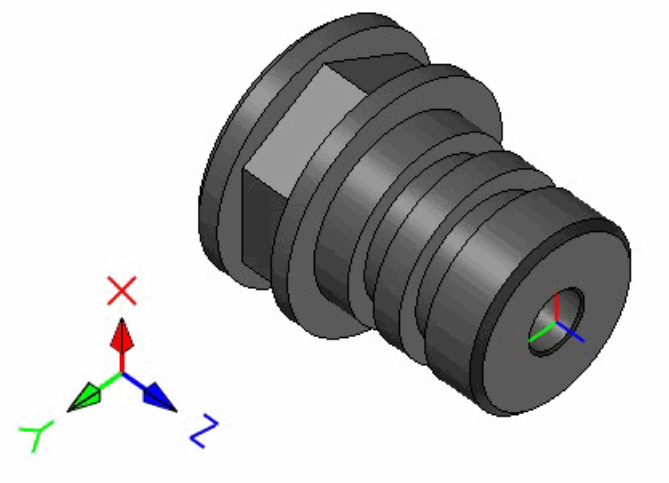
This section describes how to import a 3D solid model into PartMaker and extract geometry from the model.

If your PartMaker installation does not allow you to import solid models, or you prefer to work in 2D, skip this section and follow the instructions in Importing a 2D files (see page 10).

1 Select **File > Import > X_T Parasolid Text File** to display the **Import Parasolid X_T File** dialog.

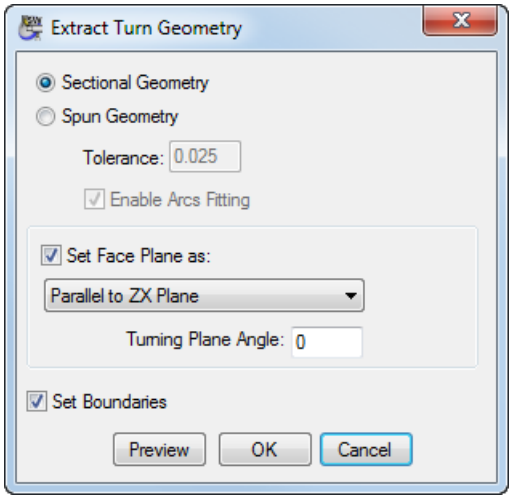
In the Getting Started folder of your PartMaker installation, select the getting_started_metric.x_t file and click **Open**.

PartMaker displays the solid model in the 3D area of the graphics window:

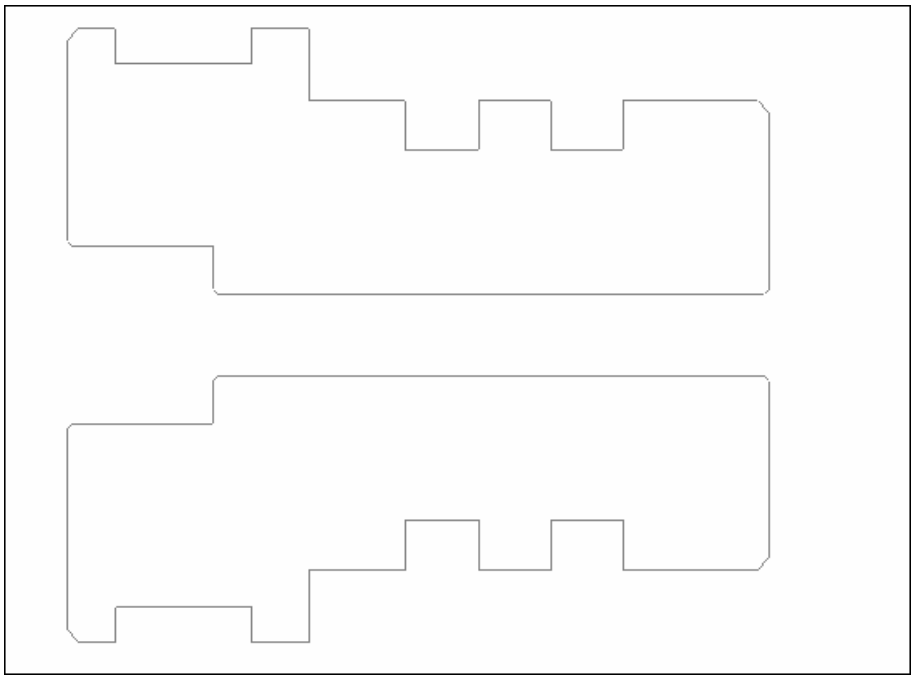


2 Click the **Extract Turn Geometry** button on the Solids toolbar to display the **Extract Turn Geometry** dialog.

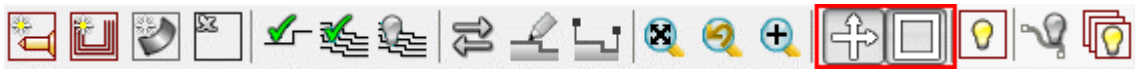
3 Complete **Extract Turn Geometry** dialog as shown, then click **OK**.

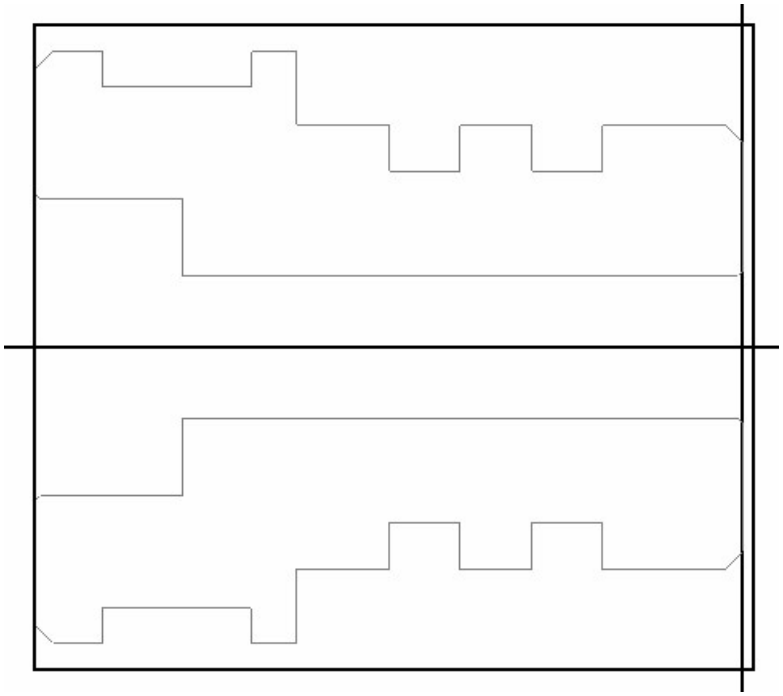


PartMaker displays a copy of the 2D profile in the Face window:



4 Click the **Show Axes** and **Show Boundaries** buttons to display axes and boundaries in the PartMaker window:





Importing a 2D geometry file

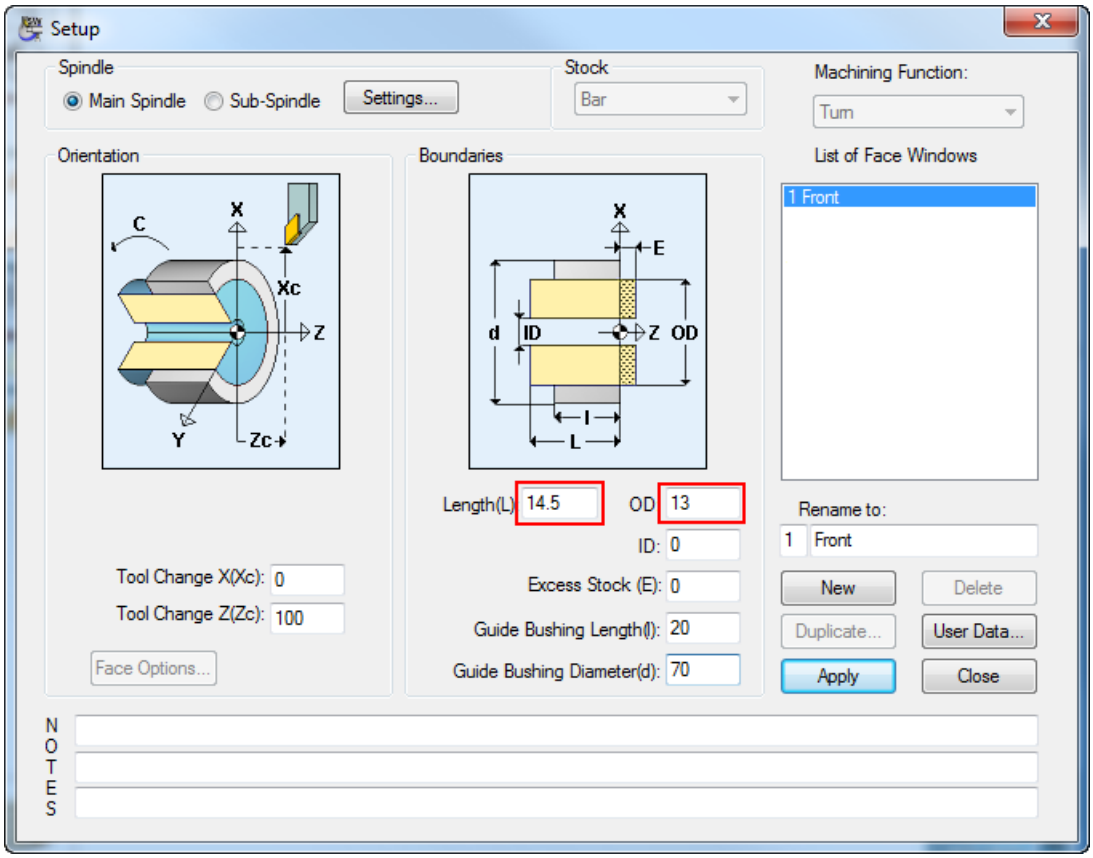
This section describes how to import a 2D .DXF file containing part geometry into PartMaker. You need to perform these steps only if your PartMaker installation does not allow you to import solid models, or if you prefer to work in 2D.

1 Select **File > Import > DXF File** to display the **Import DXF File** dialog.

In the Getting Started folder of your PartMaker installation, select the turn_profile_metric.dxf file and click **Open**.

PartMaker displays the 2D geometry in the PartMaker window.

2 Select **View > Setup** to display the **Setup** dialog and change the **Length** and **OD** values to those shown below:

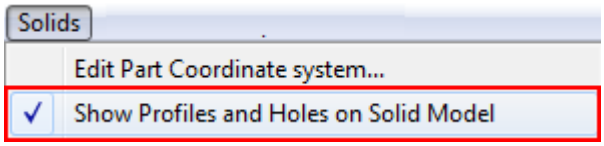


3 Click **Close**.

Programming and verifying toolpaths

This section describes how to program and verify the toolpaths required to machine the part.

If you are using an imported 3D model, check that the **Solids > Show Profiles and Holes on Solid Model** option is selected so you will be able to view toolpath profiles on the solid model:



Important programming convention

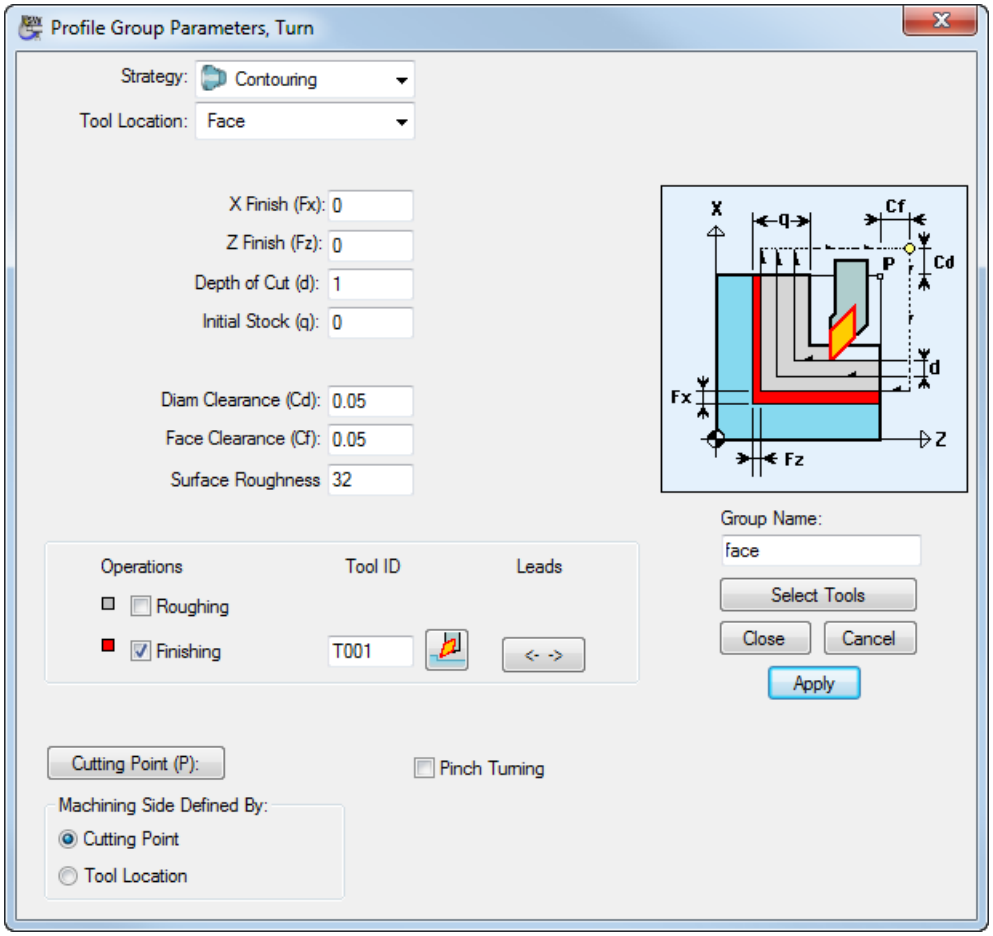
When creating geometry, setting up tools and programming toolpaths in PartMaker/SwissCAM, it is important to remember that PartMaker assumes that all work is done moving from right to left in the Z- axis (that is, it assumes that the collet or chuck is always on the left). This convention applies regardless of the physical construction of the machine. PartMaker makes sure that the NC Program is created in the required coordinate system for your machine during the postprocessing process.

Creating a turning profile group – facing

This section describes how to create a profile group for the facing operation.

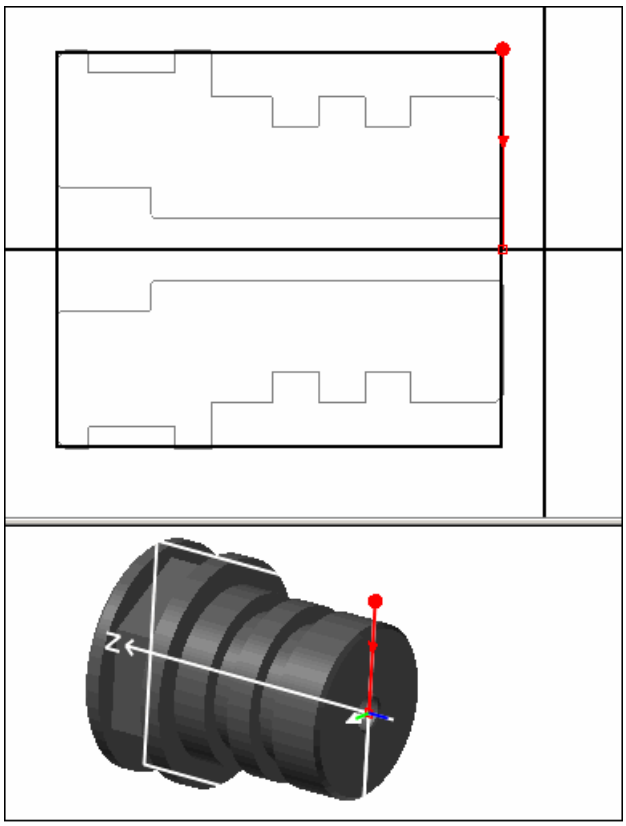
1 Click the **New Profile Group** button to display the **Profile Group Parameters, Turn** dialog.

2 Complete the **Profile Group Parameters, Turn** dialog, as shown, then click **Close**.



*To complete the Finishing Tool ID field, click **Select Tools** and select T001 on the **Select Tool** dialog. Clicking **Select** closes the **Select Tool** dialog and returns you to the **Profile Group Parameters, Turn** dialog.*

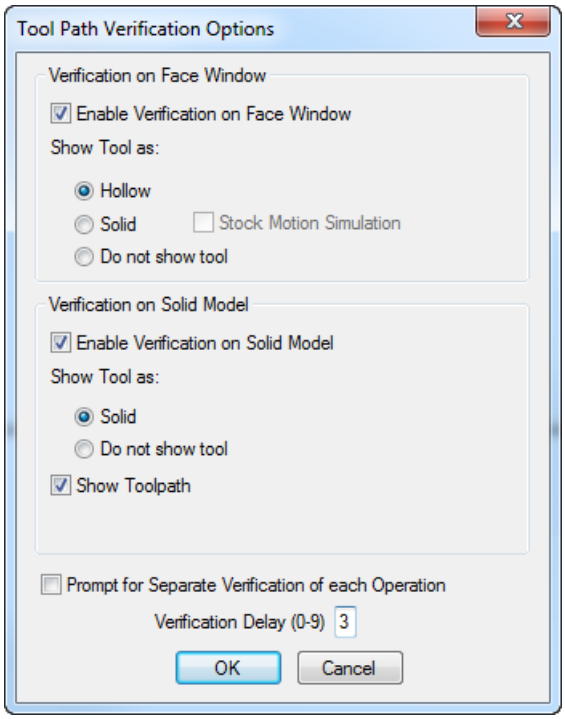
Because you selected a **Tool Location** of *Face*, PartMaker automatically completes the profile by applying the facing toolpath to the part geometry and the 3D model:



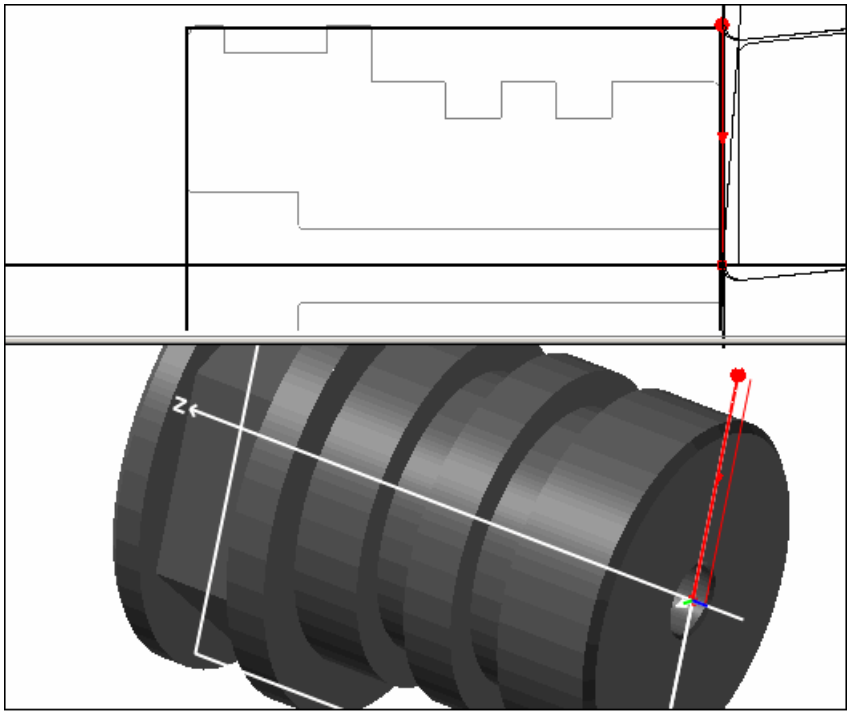
In the toolpath, the circle indicates the start point; the arrow indicates the direction of travel; and the square represents the next 'point' to be cut (which, in this case, is also the last point).

3 To check the accuracy of the cut you have defined, click the **Verify Work Group Toolpath** button to display the **Tool Path Verification Options** dialog.

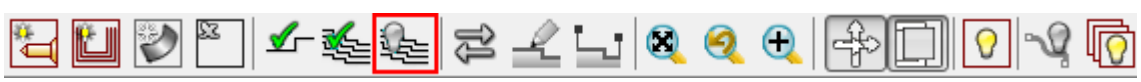
4 Complete the **Tool Path Verification Options** dialog as shown, then click **OK**.



PartMaker displays a 2D representation of the tool moving across the face of the part and display results in the PartMaker window:



5 Click the **Hide Every Toolpath** button to remove the verification details from the display.



Creating a turning profile group – outside diameter

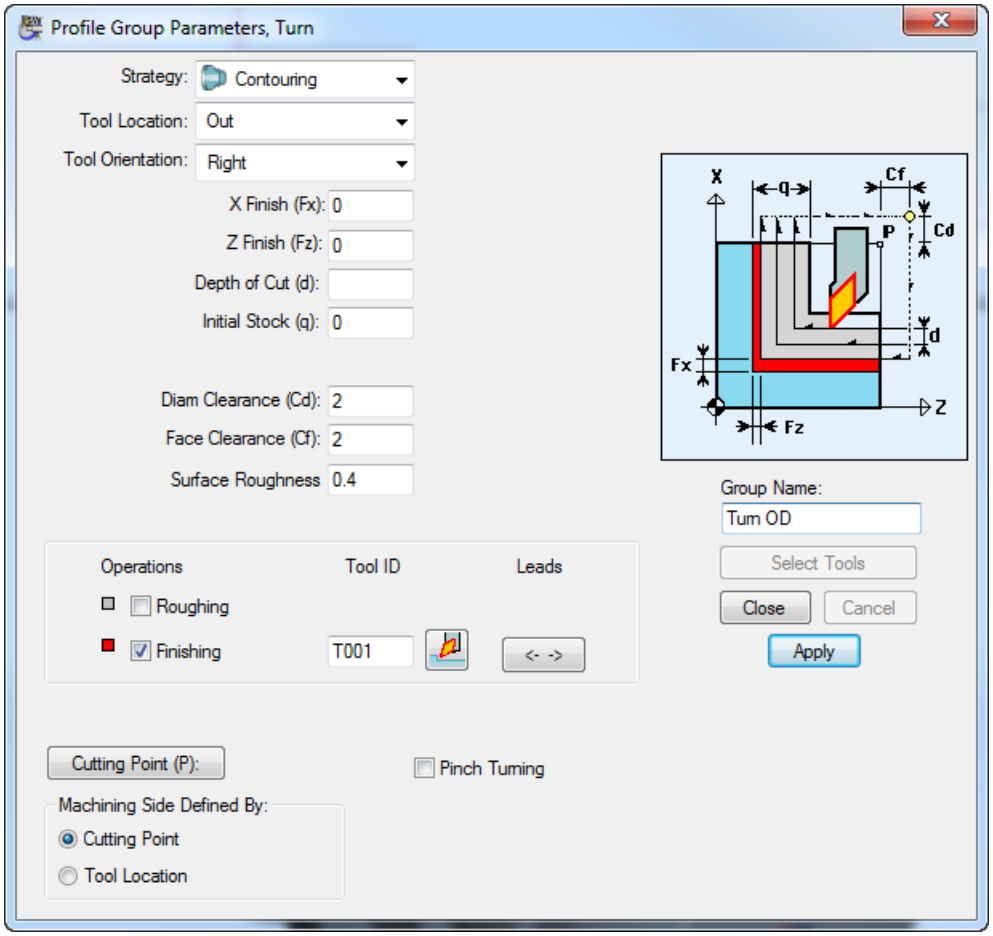
This section describes how to create a profile group for the outside diameter of the part:

1. Select the color you want to use for this toolpath from the Color Palette:

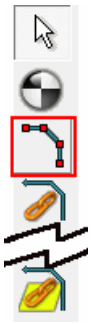
   Using different colors for your toolpaths makes it easier to identify the toolpaths you have programmed when viewing the part geometry, or the solid model, in the PartMaker window.

2. Click the **New Profile Group** button to display the **Profile Group Parameters, Turn** dialog.

3. Complete the **Profile Group Parameters, Turn** dialog as shown, then click **Close**.



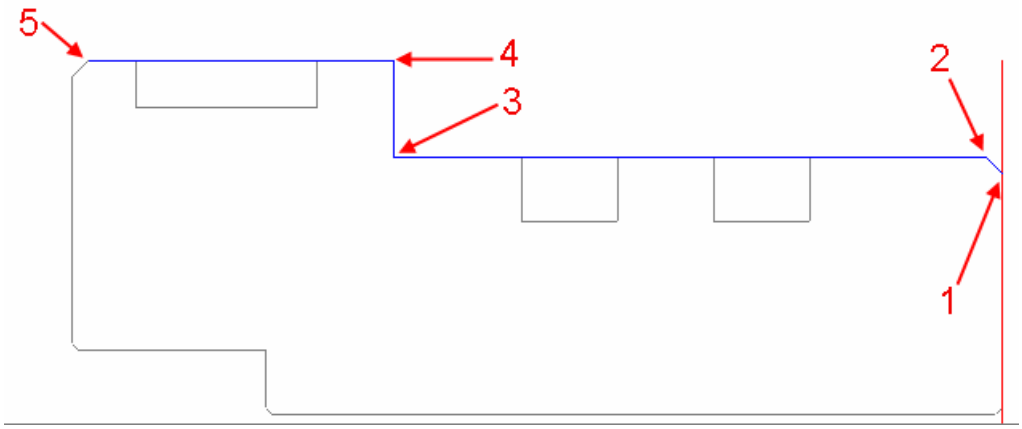
4 Click the **Define Profile** button, so you can specify where the profile goes on the part.



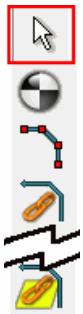
PartMaker automatically defines profiles for facing and cut-off operations. For other operations, you need to define the profile.

5 Click the **End of an Element Snap Mode** button to specify that you want to define the profile by using the part's geometry as 'snap points' for the toolpath:

6 Click on the part geometry to specify the following snap points in the order shown:

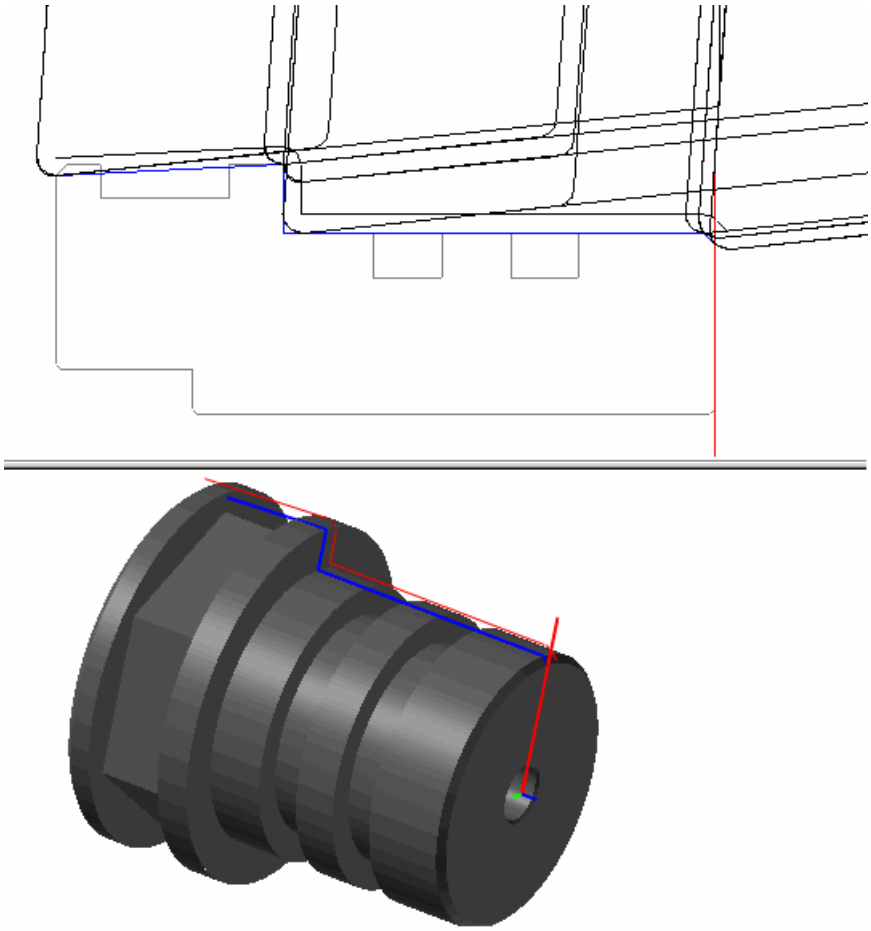


7 Click the **Selection** button to save the profile.

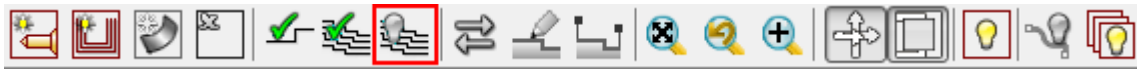


8 To verify the toolpath click the **Verify Work Group Toolpath** button, then click **OK** on the **Tool Path Verification Options** dialog.

You can now visualize the toolpath:

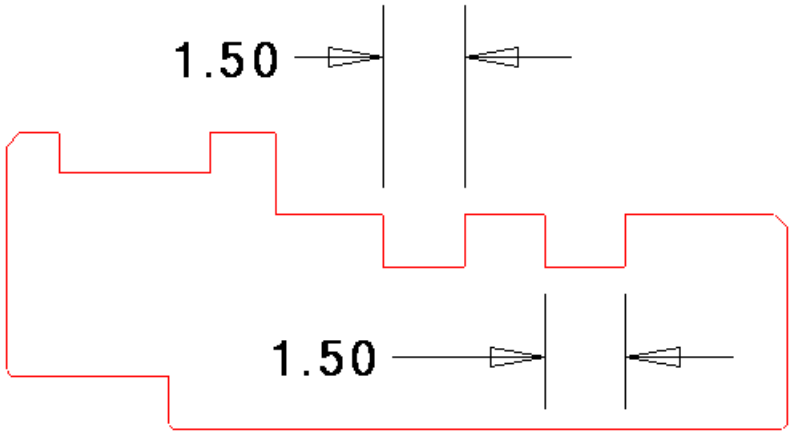


9 Click the **Hide Every Toolpath** button to remove the verification details from the display.



Creating a turning profile group – grooves

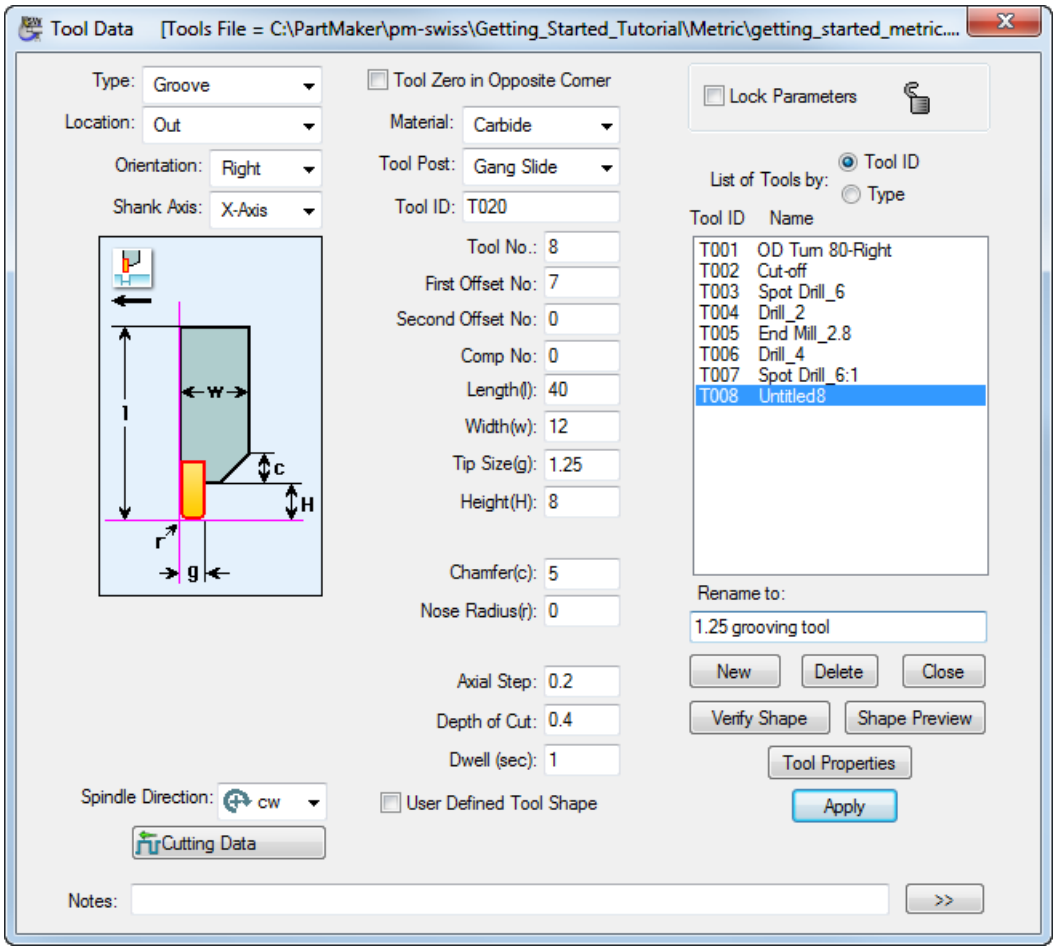
This section describes how to program the two grooves on the turned diameter of the part:



Before programming this toolpath, you must create a new tool, as no suitable tool currently exists in the Tools database.

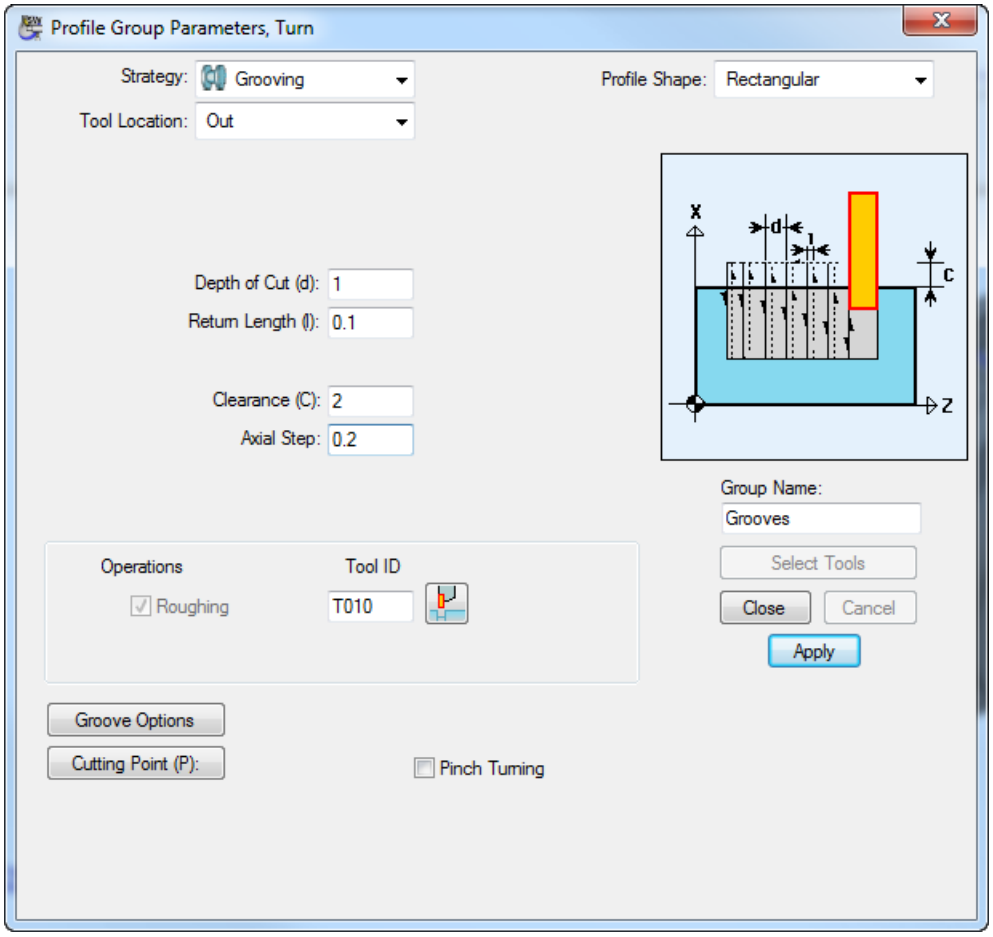
1 Click the **Tools** button to display the **Tool Data** dialog.

2 On the **Tool Data** dialog, click **New,** then complete the dialog as shown:



3 Select a colour for the toolpath from the Color Palette, then click the **New Profile Group** button to display the **Profile Group Parameters, Turn** dialog.

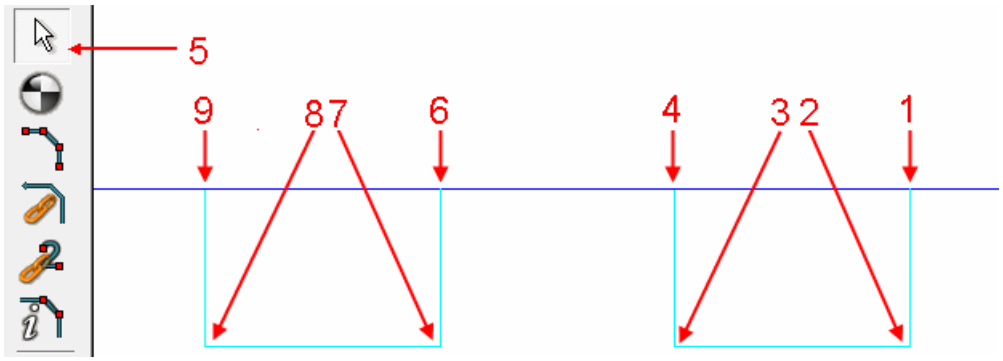
4 Complete the **Profile Group Parameters, Turn** dialog, as shown:



*Use the **Select Tools** button to select the new tool you created in Step 2.*

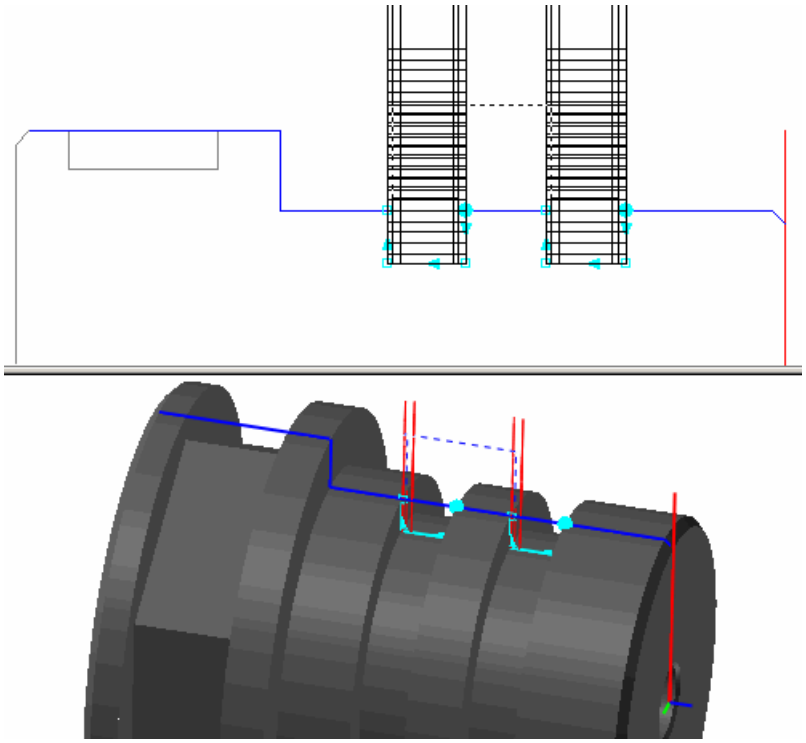
5 Click the **Define Profile** button, then click the **End of an Element Snap Mode** button:

6 You can now define the profile using snap points. Follow the order shown below, including clicking the **Selection** button (see 5) when you have finished the first profile, to 'break' the profiles into two distinct pieces:

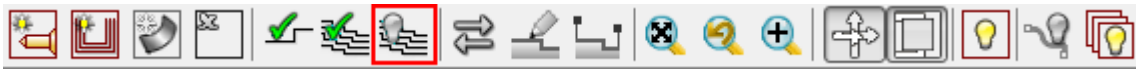


7 To verify the toolpath, click the **Verify Work Group Toolpath** button, then click **OK** on the **Tool Path Verification Options** dialog.

You can now visualize the toolpath:



8 Click the **Hide Every Toolpath** button to remove the verification details from the display.

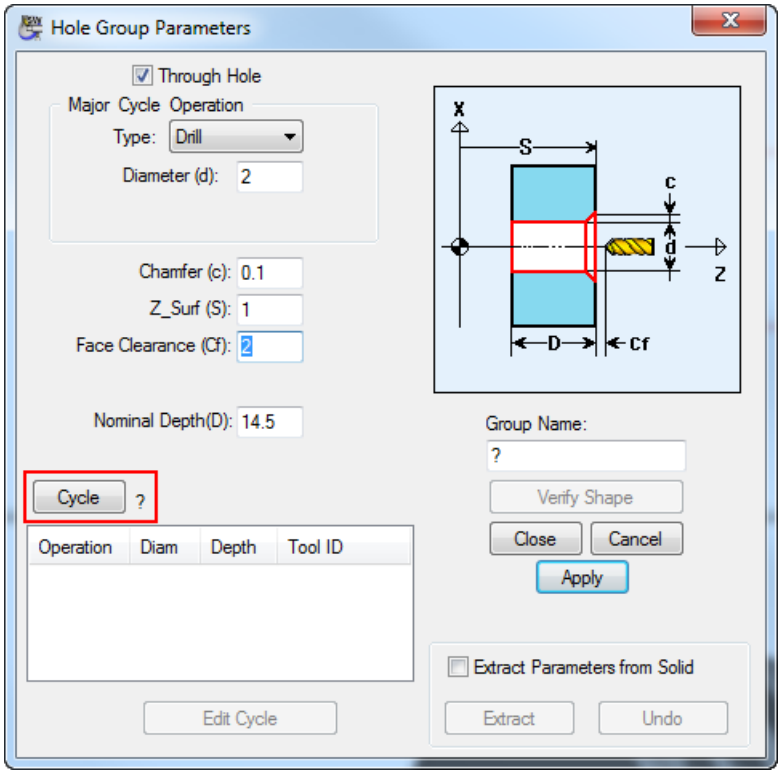


Creating a turning hole group – through hole

This section describes how to program the hole through the center of the part:

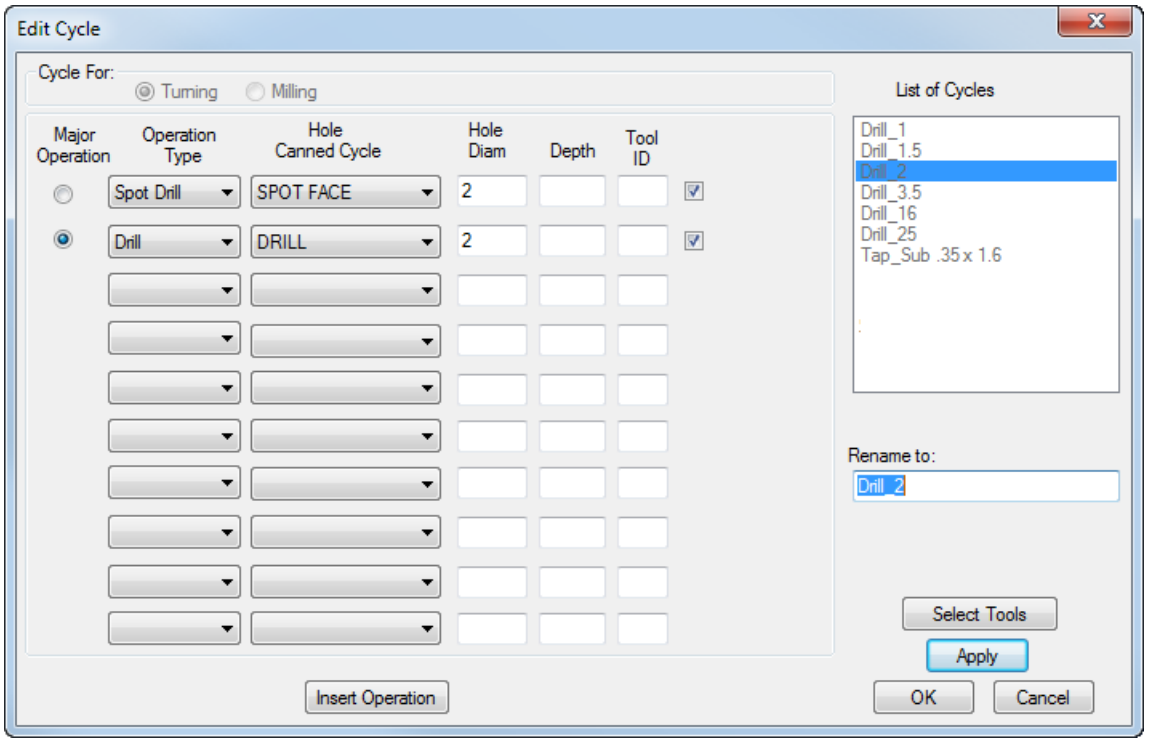
1. Select a colour for the toolpath from the Color Palette, then click the **New Hole Group** button to display the **Hole Group Parameters** dialog.

2. Complete the **Hole Group Parameters** dialog as shown, then click **Cycle** to display the **Cycle** dialog.

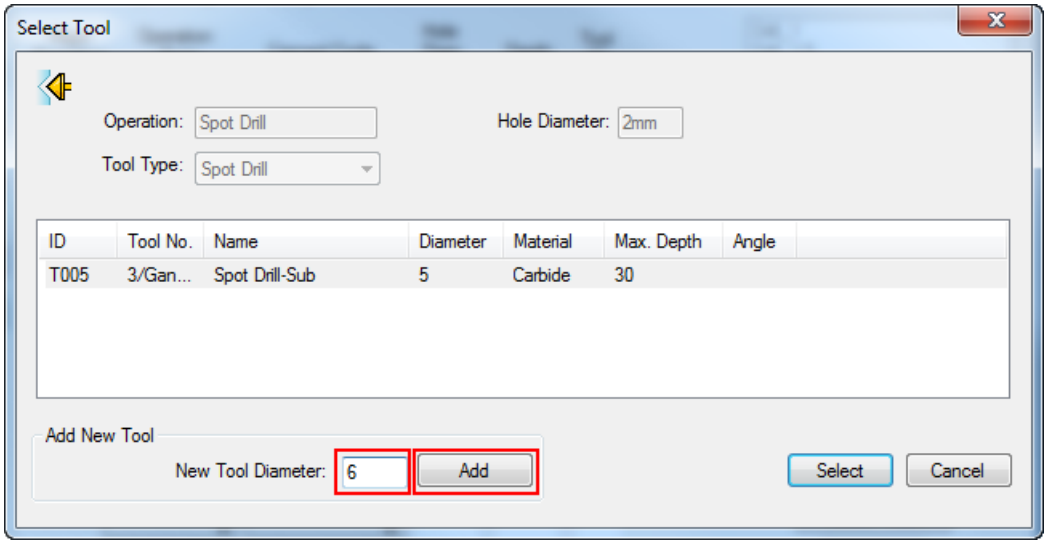


3. On the **Select Cycle** dialog, click **Add New Cycle** to display the **Edit Cycle** dialog.

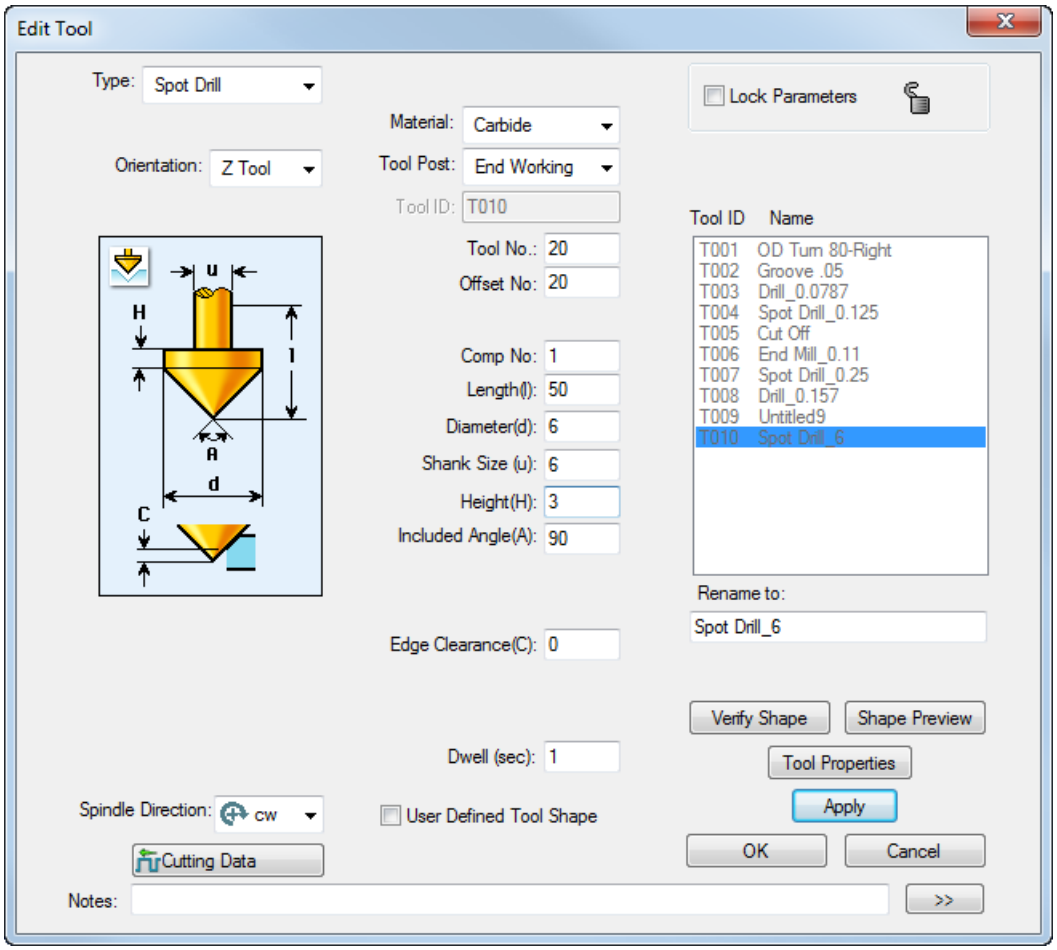
4 Click **Insert operation**, then complete the **Edit Cycle** dialog as shown:



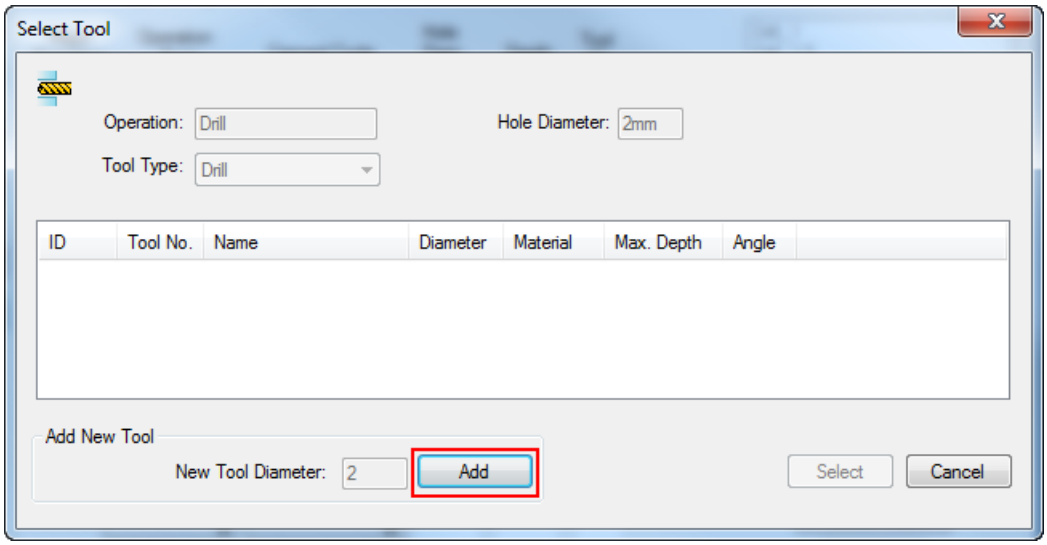
5 Click **Select Tools**, change the **New Tool Diameter** to **6** and then click **Add**.

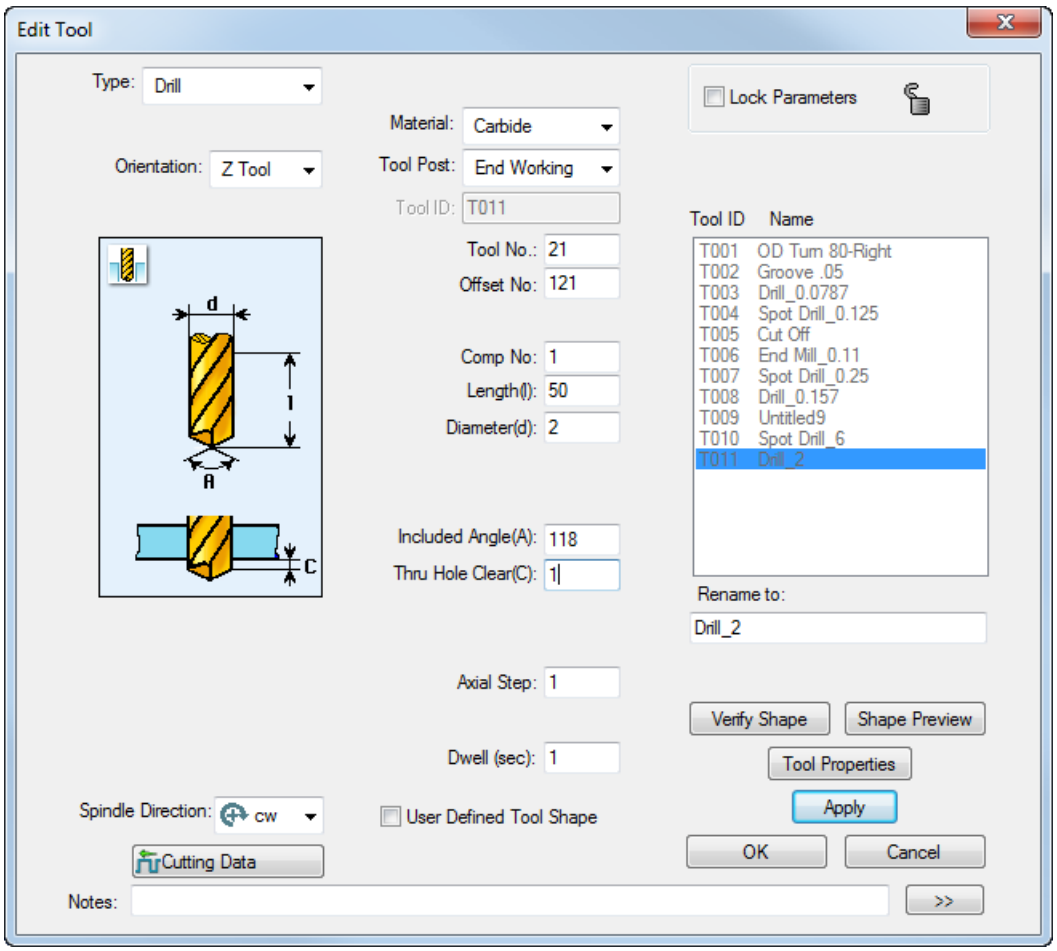


6 Complete the **Edit Tool** dialog, as shown, noticing that the **Orientation** and **Tool Post** values have changed. Click **OK** when you have finished.

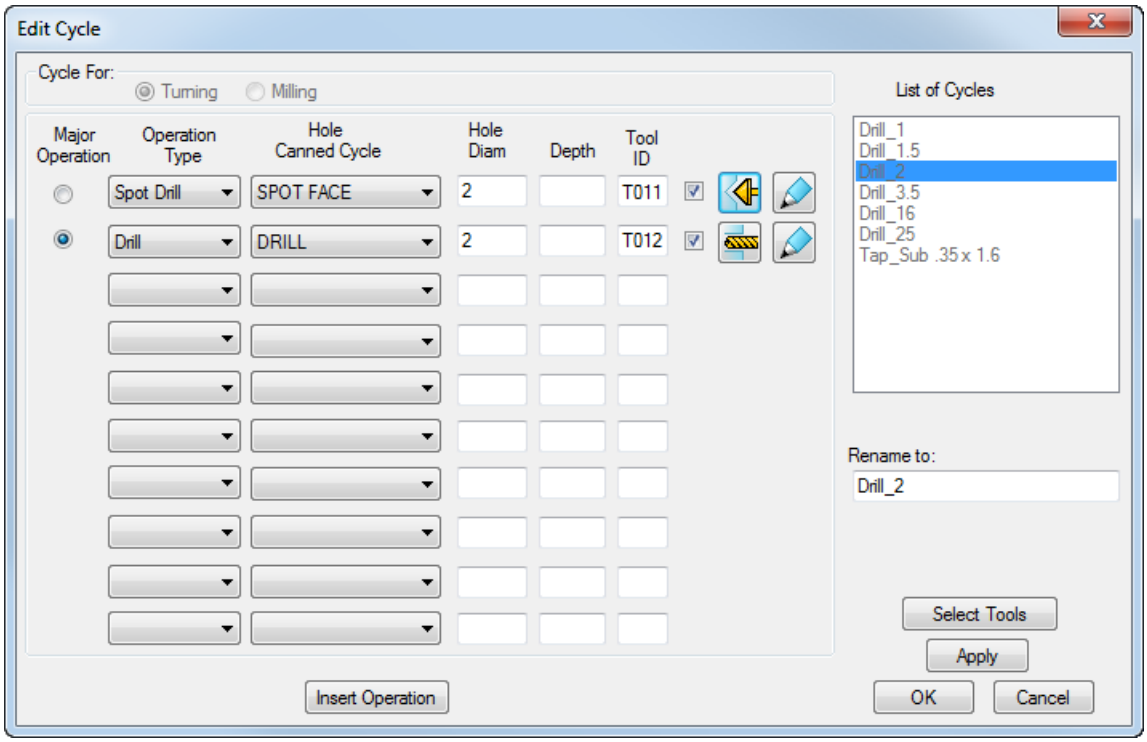


7 You now need to create a new drill tool suitable for the diameter of this hole To do this, click **Add** on the **Select Tool** dialog, then complete the **Edit Tool** dialog as shown:

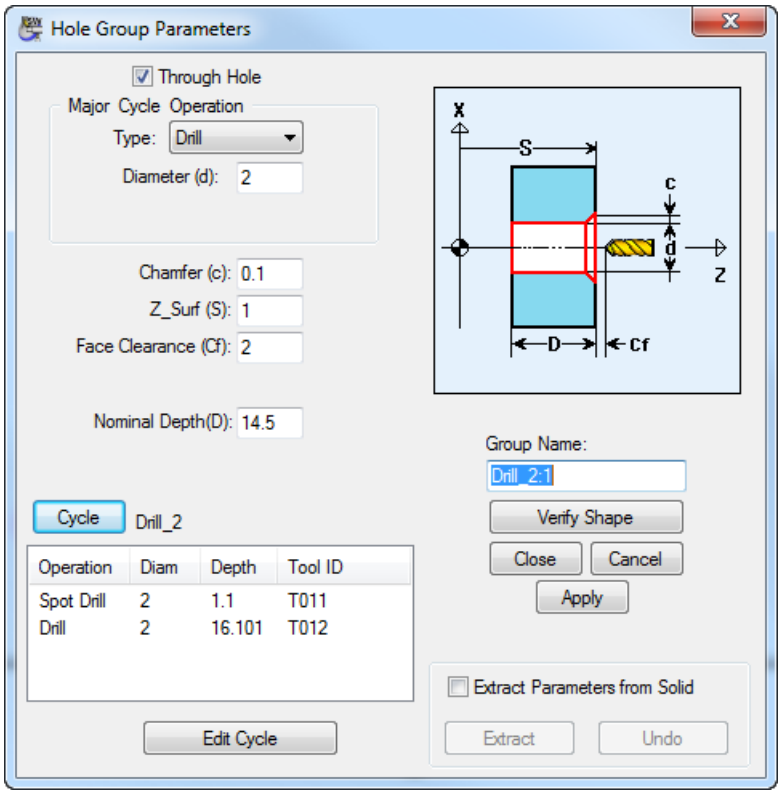




8 When you have completed the **Edit Tool** dialog, click **OK** to confirm your changes. PartMaker returns to the **Edit Cycle** dialog:

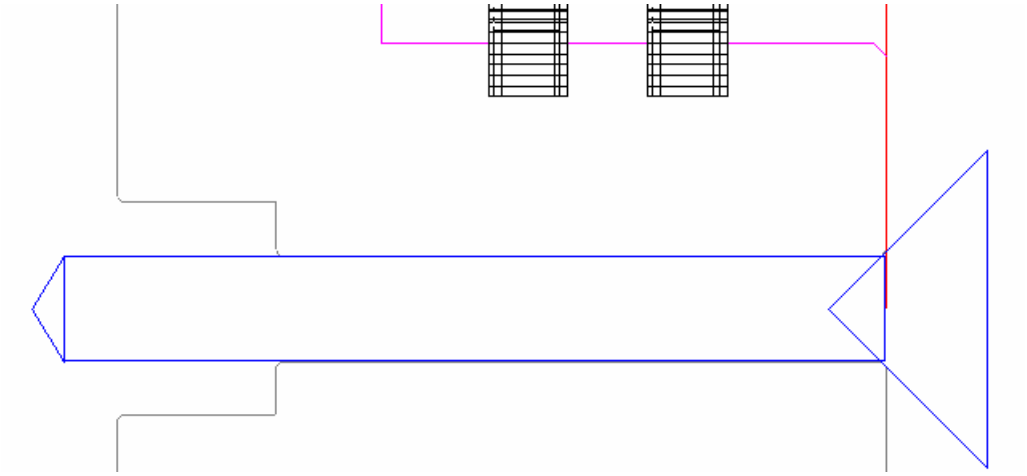


9 Click **OK** on the **Edit Cycle** dialog to return to the **Hole Group Parameters** dialog:



10 On the **Hole Group Parameters** dialog, click **Apply,** then **Close.**

You can now visualize the toolpath:

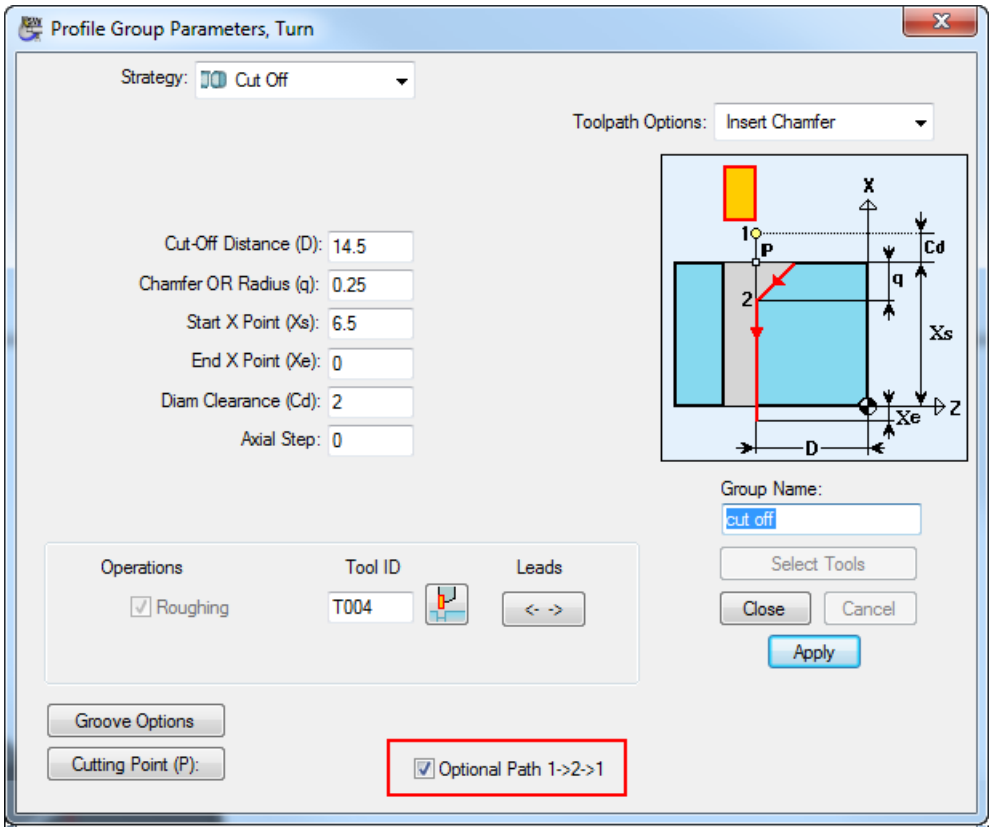


Creating a turning cut-off toolpath

This section describes how to create a toolpath for the cut-off operation.

1 Select a colour for the toolpath from the Color Palette, then click the **New Profile Group** button.

2 Complete the **Profile Group Parameters, Turn** dialog, making sure **Optional Path 1>-2->1** is selected, then click **Close**.



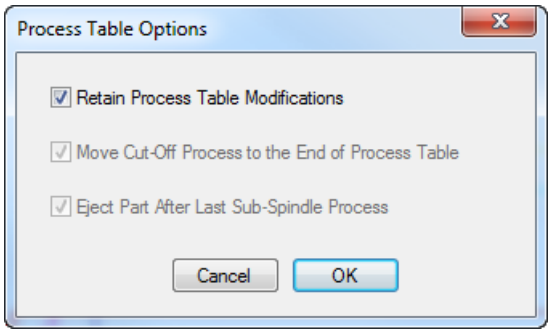
PartMaker automatically completes the profile by applying the cut-off toolpath to the part geometry and the 3D model.

Simulating the turning toolpaths

This section describes how to view a 3D simulation, which shows how the toolpaths will be machined. Now you have created all the turning profile groups for the part, this is a good time to run a simulation.

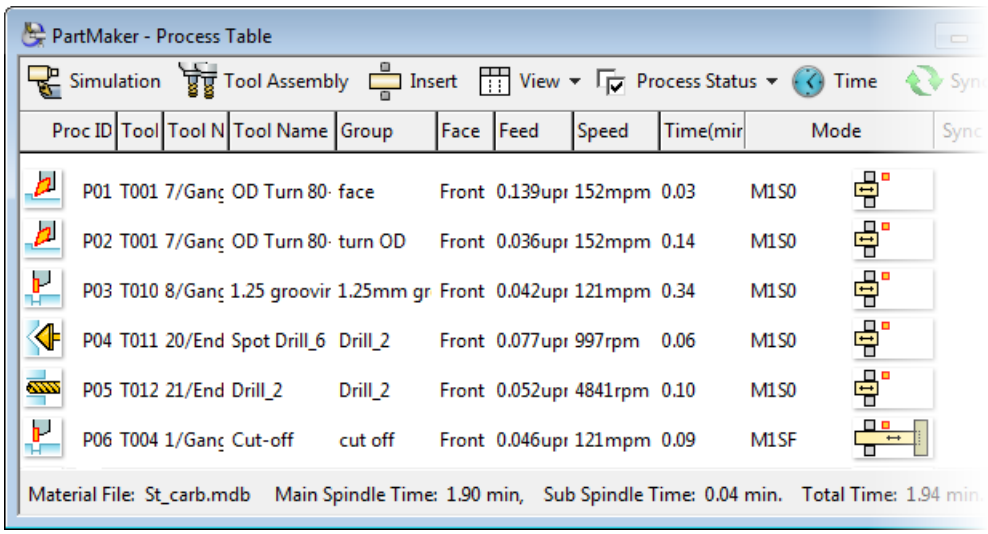
1 Click the **Generate Process Table** button to create a Process Table, which lists the processes required to machine the toolpaths.

2 Complete the **Process Table Options** dialog as shown, then click **OK**.

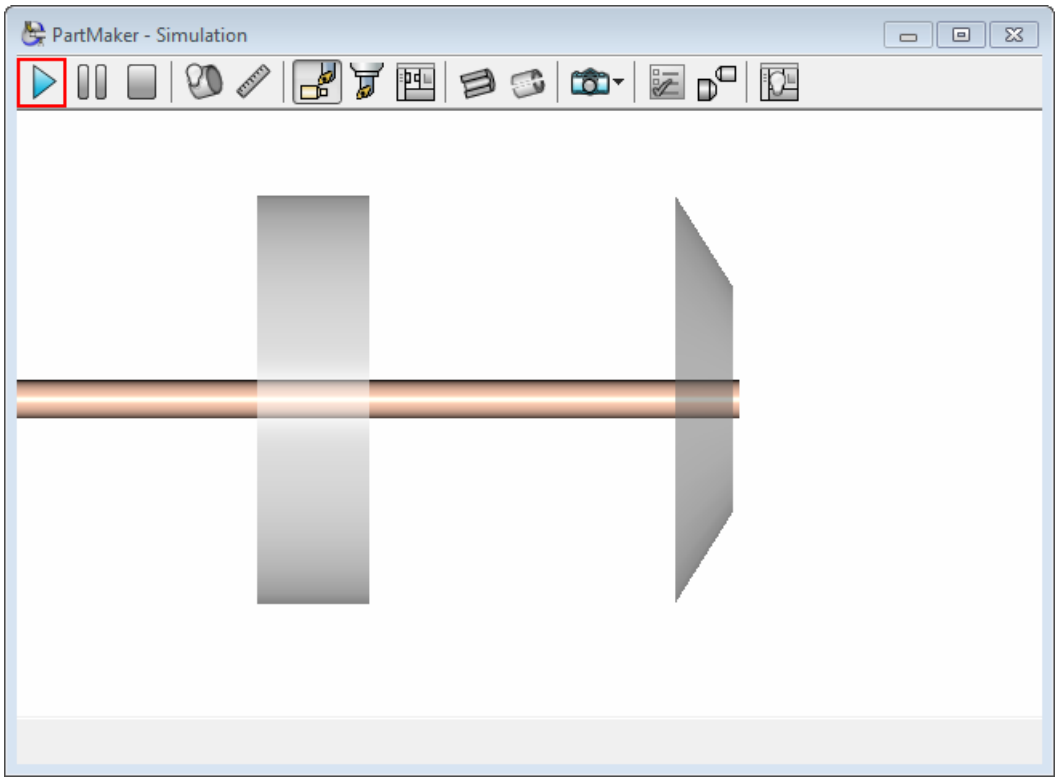


*If PartMaker displays a warning that no ejection is programmed, click **Yes** to continue.*

3 View the Process Table, then click Simulation.



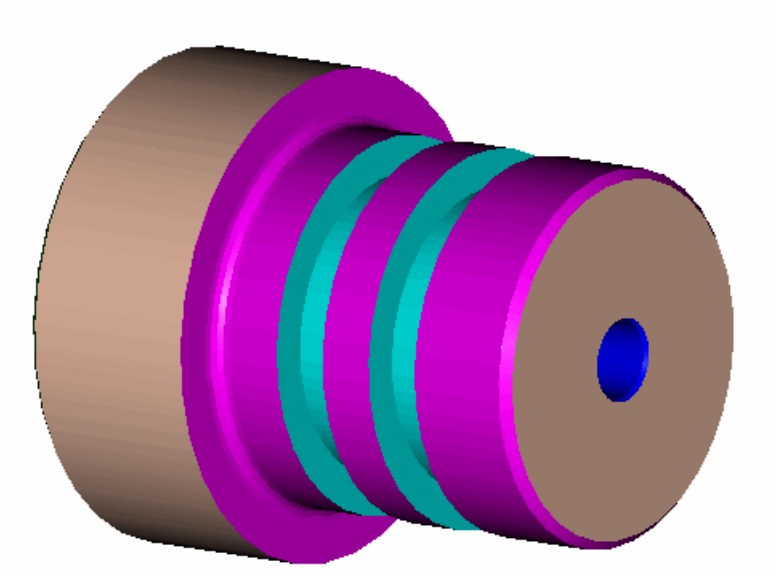
4 In the **Simulation** window, click **Play** to start the simulation.



*If <PRODUCT, displays a warning that no eject operation is programmed, click **Yes** to continue.*

5 When the simulation is complete, click the **Show Finished Part** button.

PartMaker displays a 3D representation of the finished part:



Creating a milling profile group – hexagon

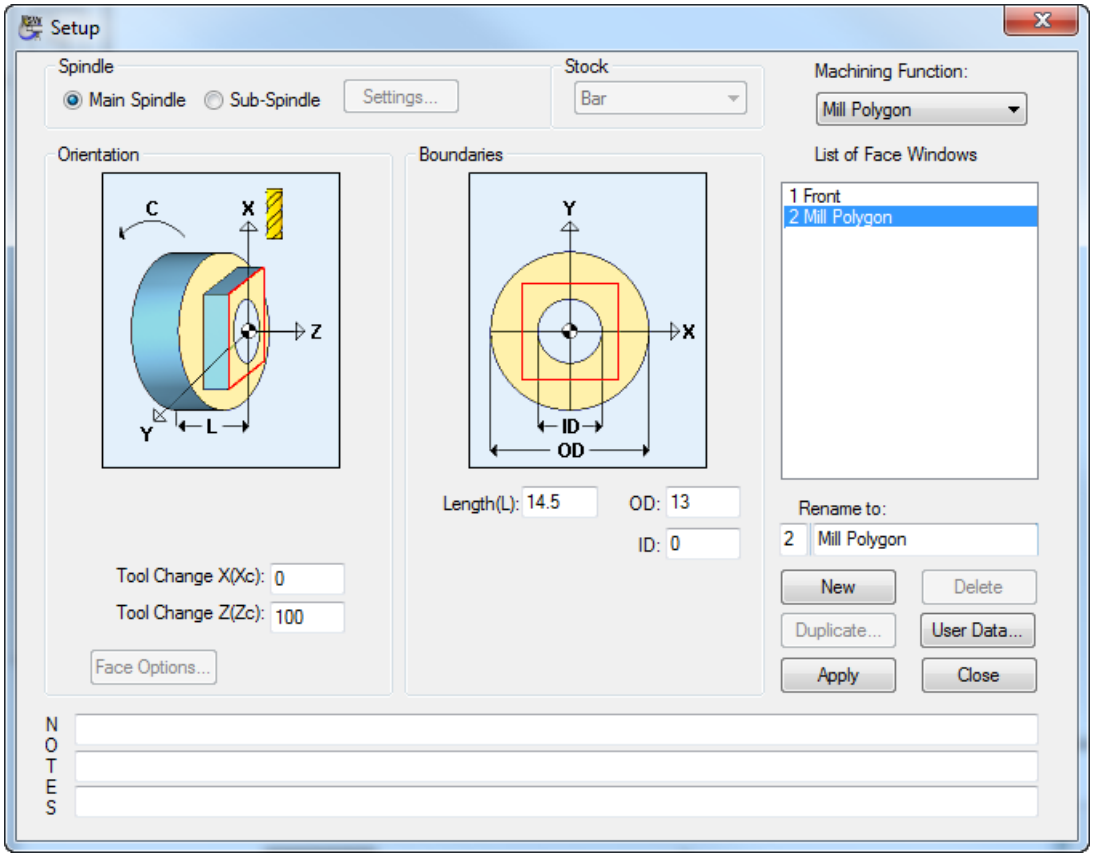
This section describes how to create a milling toolpath to mill the hexagon on the part.

This section shows you how to create the toolpath using geometry extracted from a 3D solid model. If, however, you want to use geometry from an imported 2D .DXF file instead, select **File > Import > DXF File** to display the **Import DXF File** dialog and then select the hex_profile_metric.dxf file from the Getting Started folder of your PartMaker installation. PartMaker displays the 2D geometry in the PartMaker window. You can then follow the steps described in this section.

So far, you have created turning toolpaths in the **Front** face window. To create a toolpath that uses a different machining function (milling), you need to create a new Face window.

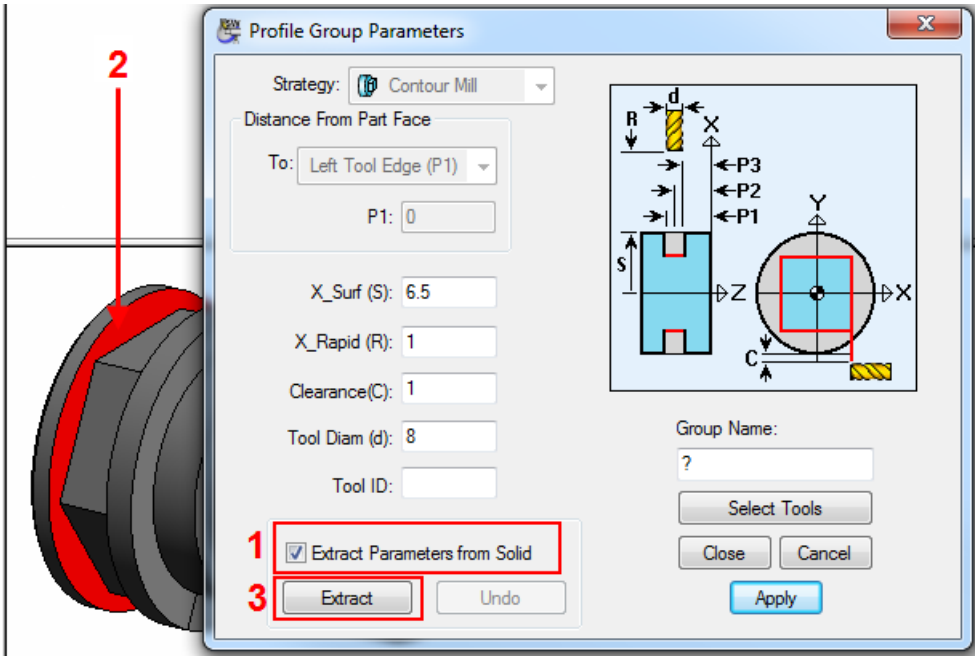
1 Select **View > Setup** and then click **New**.

2 Complete the **Setup** dialog as shown, then click **Close** to finish.

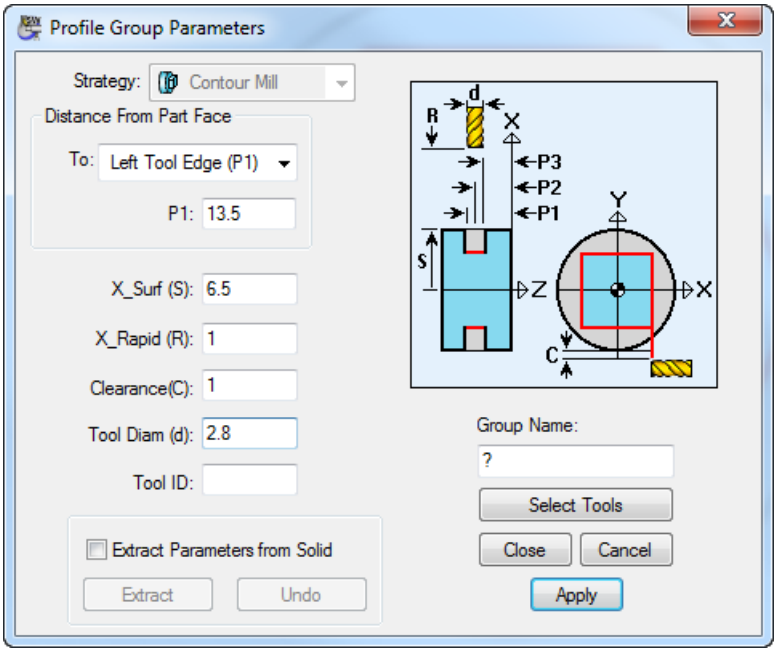


3 Click the **New Profile Group** button on the toolbar to display the **Profile Group Parameters** dialog.

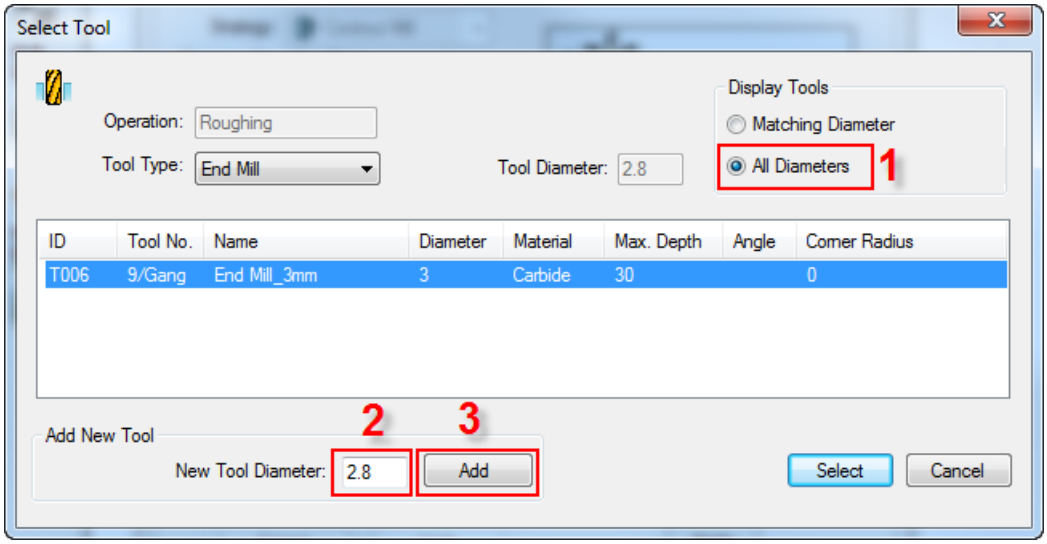
- If you are using the imported 3D solid model, complete the dialog as shown, by selecting the **Extract Parameters From Solid** option, selecting the plane in the Solids window and then clicking **Extract**.



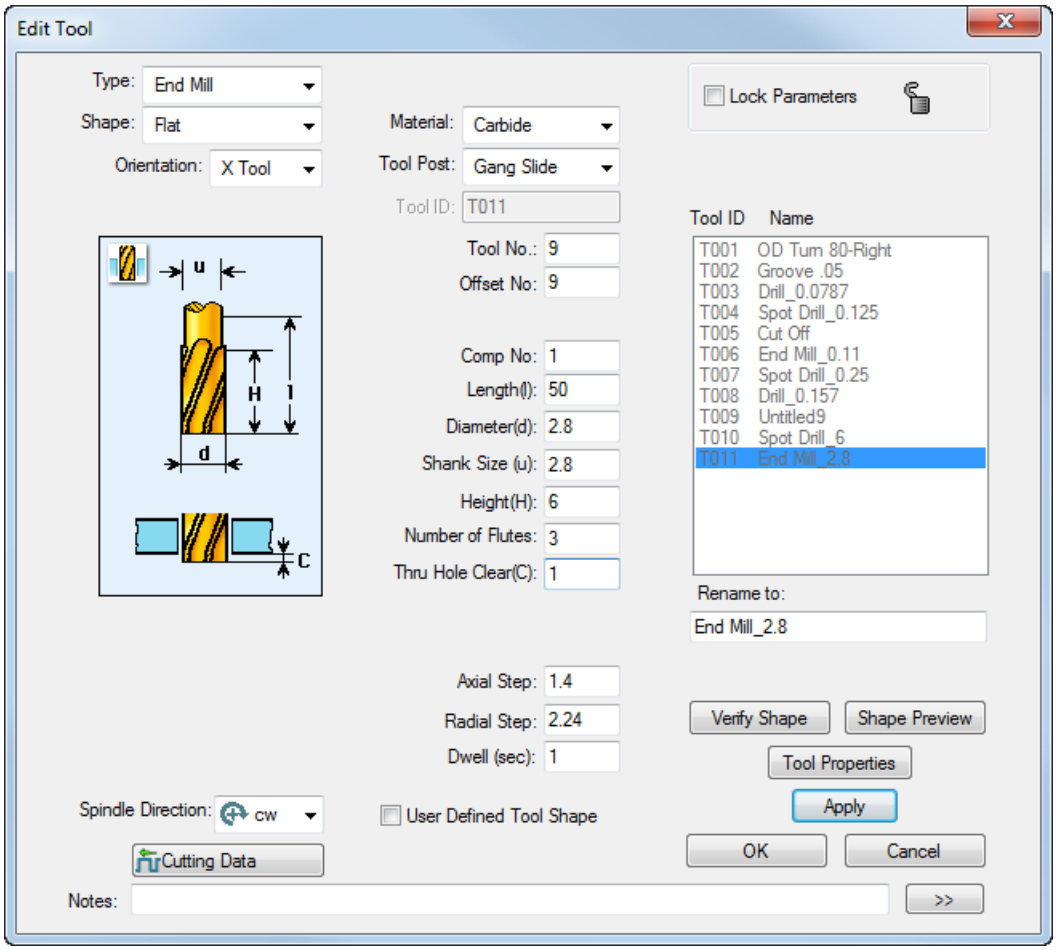
- If you are using the imported hex_profile_metric.dxf file, complete the dialog as shown:



4 Click **Select Tools** to display the **Select Tool** dialog, then select **All Diameters**. Notice that only a 3mm milling cutter is available, which is too wide for the 2.8mm hexagonal slot. To specify that you want to create a new 2.8mm tool, complete the **New Tool Diameter** field as shown below, then click **Add**.

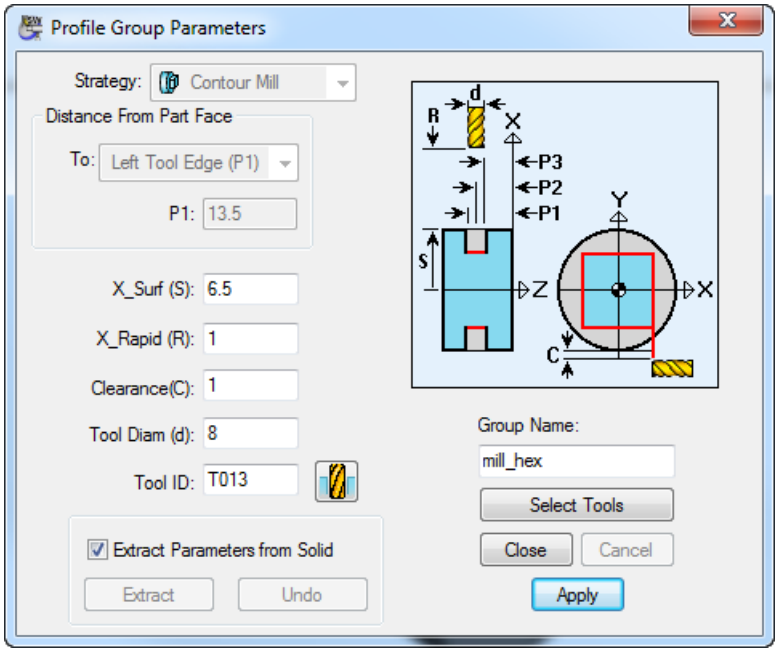


5 To create a 2.8mm milling tool, complete the **Edit Tool** dialog as shown and then click **OK**.

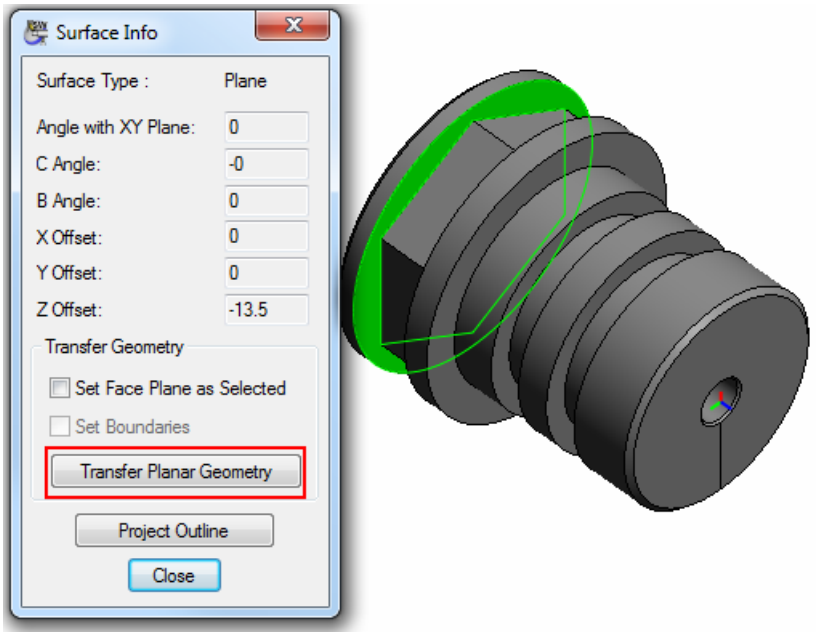


6 Check the details shown on the **Profile Group Parameters** dialog and click **Close**. If you are using an:

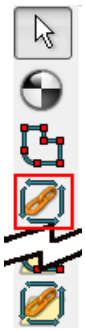
- imported 3D solid model, continue at Step 6.
- imported 2D .DXF file, continue at Step 7.



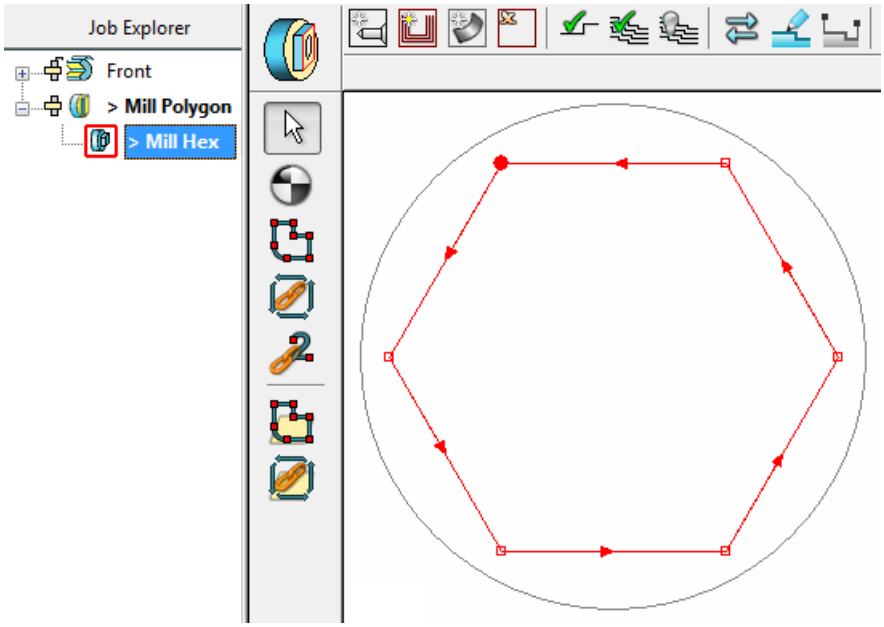
7 When using a 3D solid model, you need to transfer the planar geometry from the solid model into the 2D area of the PartMaker window before you can create the profile for that geometry. To transfer the geometry, double-click the surface on the solid model to display the **Surface Info** dialog, then click **Transfer Planar Geometry**.



8 Click the **Chain Geometry** button in the Profile toolbar to specify that you want to create the profile by selecting the end point of the hexagon.

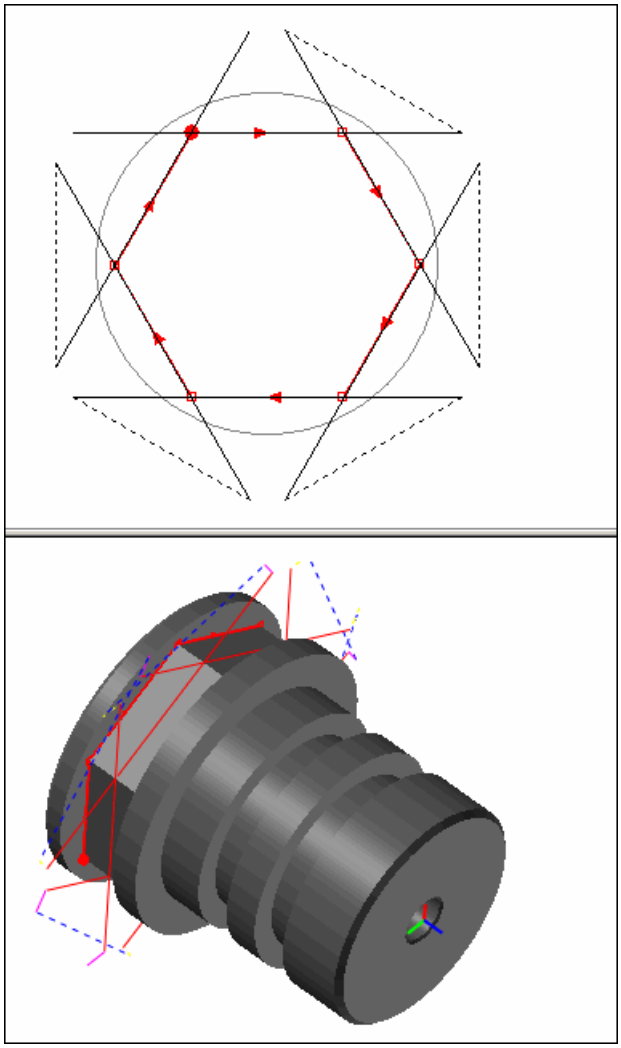


9 Click on any of the points on the hexagon to create the profile:

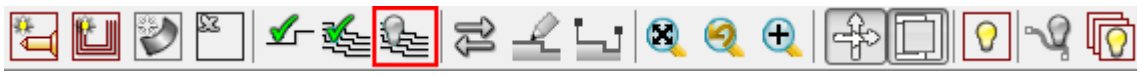


10 To verify the profile group, click the **Verify Work Group Toolpath** button, then click **OK** on the **Tool Path Verification Options** dialog.

You can now visualize the toolpath:



11 Click the **Hide Every Toolpath** button to remove the verification details from the display.



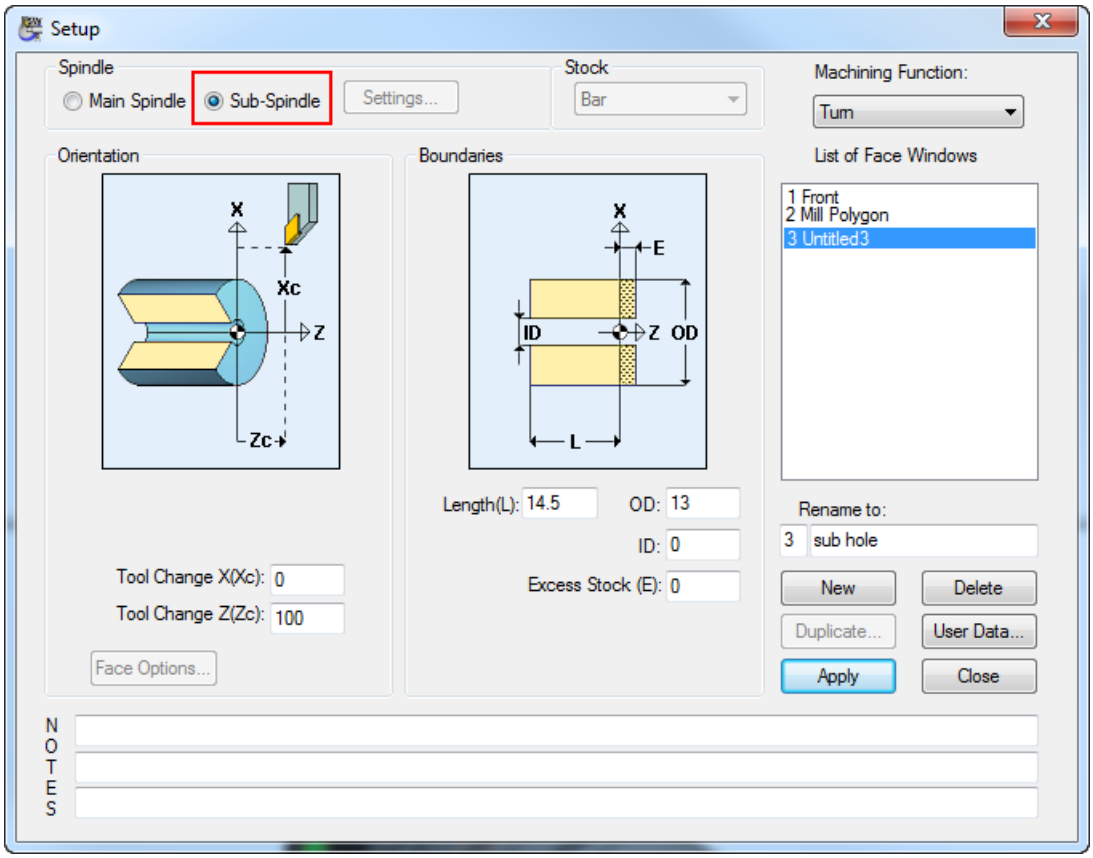
Creating a turning hole group on the sub-spindle – counterbored hole

This section describes how to program the counterbored hole on the rear of the part.

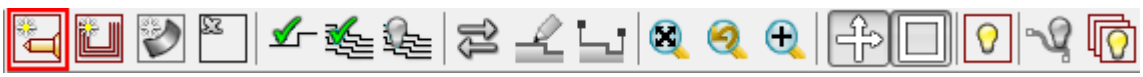
As this toolpath will be machined using the sub-spindle, you first need to create a new Face window.

1 Select **View > Setup** and then click **New**.

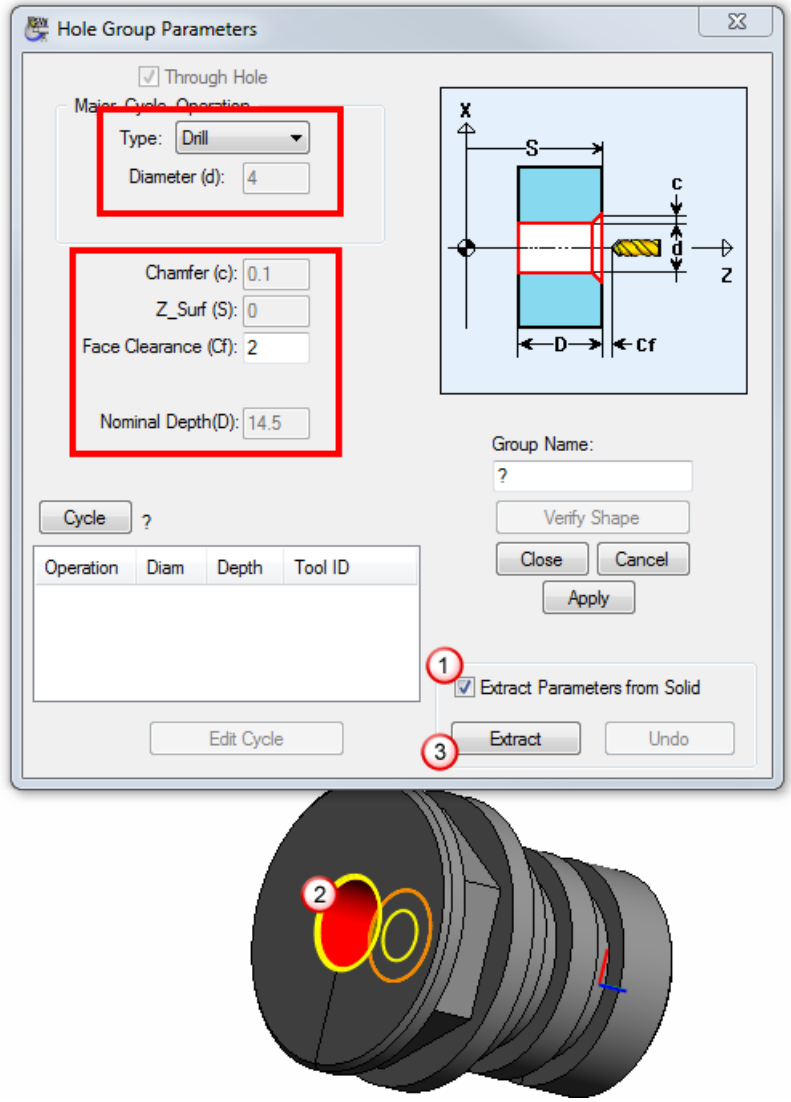
2 Complete the **Setup** dialog, as shown, remembering to select **Sub-Spindle**:



3 Select a color for the toolpath from the Color Palette, then click the **New Hole Group** button to display the **Hole Group Parameters** dialog.



4 Complete the dialog as shown, by selecting the **Extract Parameters From Solid** option, selecting the hole in the Solids window, and then clicking **Extract**.

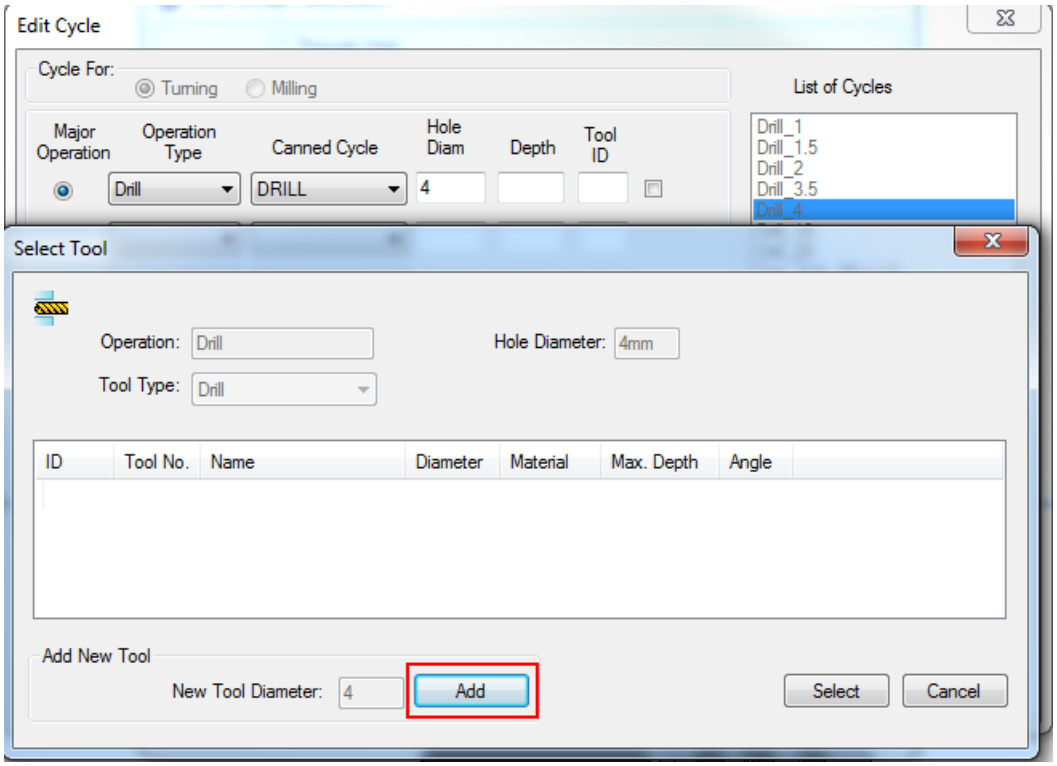


*If you are not using a solid model file and so cannot extract the hole values automatically, you need to enter the values shown in the green boxes above directly into the **Hole Group Parameters** dialog.*

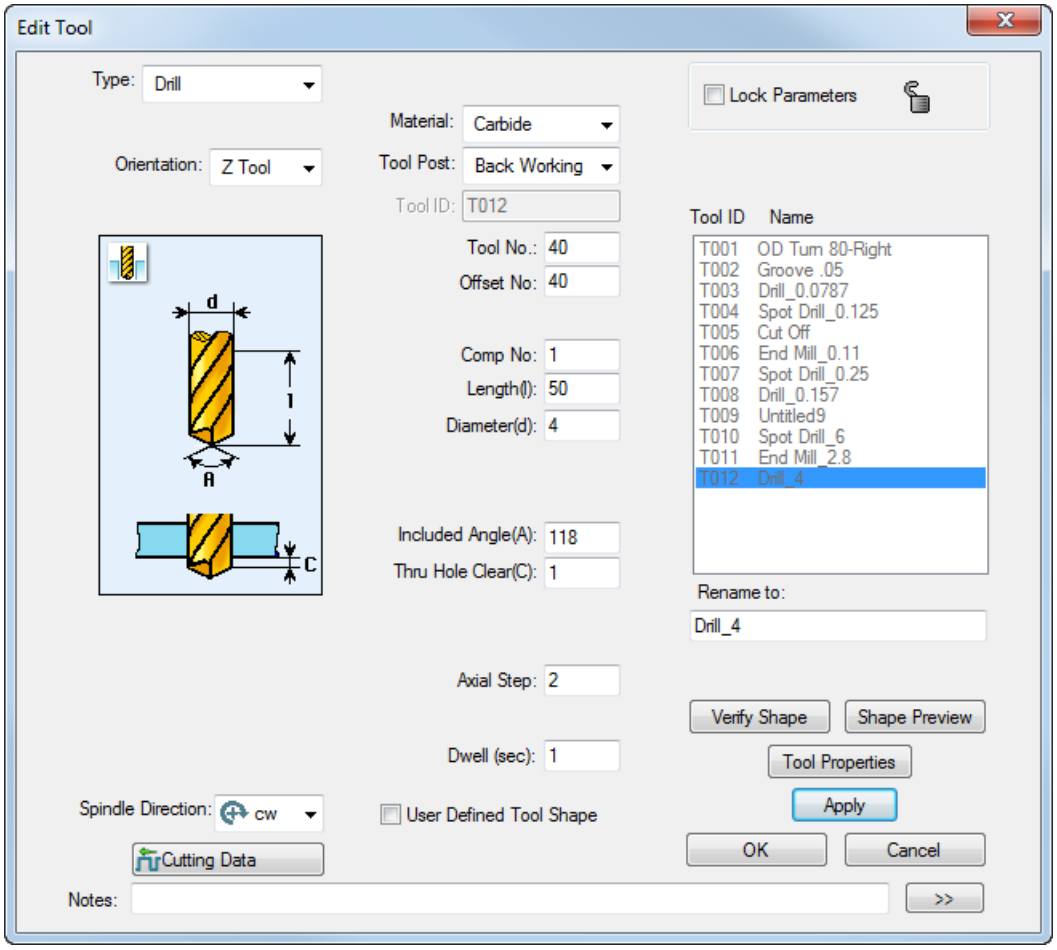
5 Click **Apply** on the **Hole Group Parameters** dialog. PartMaker displays the **Select Cycle** dialog.

6 As no suitable cycle already exists, click **Add New Cycle** to display the **Edit Cycle** dialog.

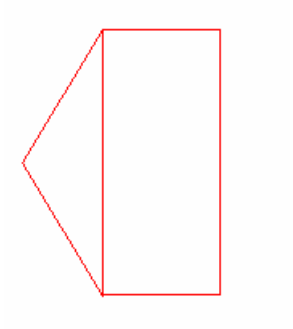
7 Click **Select Tools** to display the **Select Tools** dialog. As no suitable tool exists in the Tools database, click **Add** to add a new tool:



8 Complete the **Edit Tool** dialog as shown, then click **OK**.



9 Select **OK** to return to the **Hole Group Parameters** dialog, then click **Close**. PartMaker displays the drill toolpath in the 2D window:

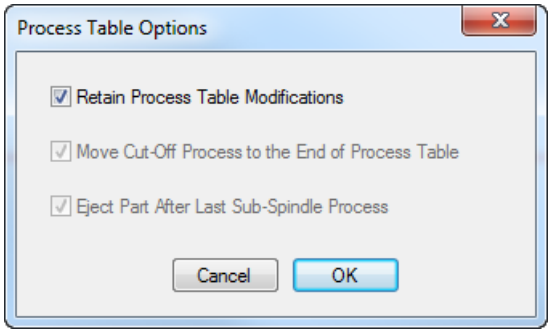


Generating the final Process Table

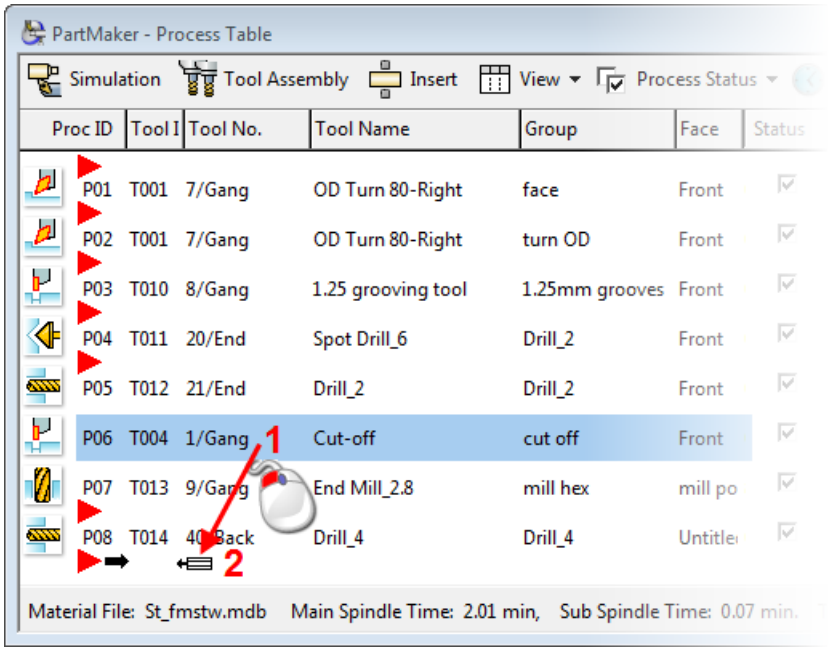
This section describes how to generate the final Process Table containing details of the processes required to machine all the toolpaths you have programmed.

1 Click the **Generate Process Table** button on the toolbar:

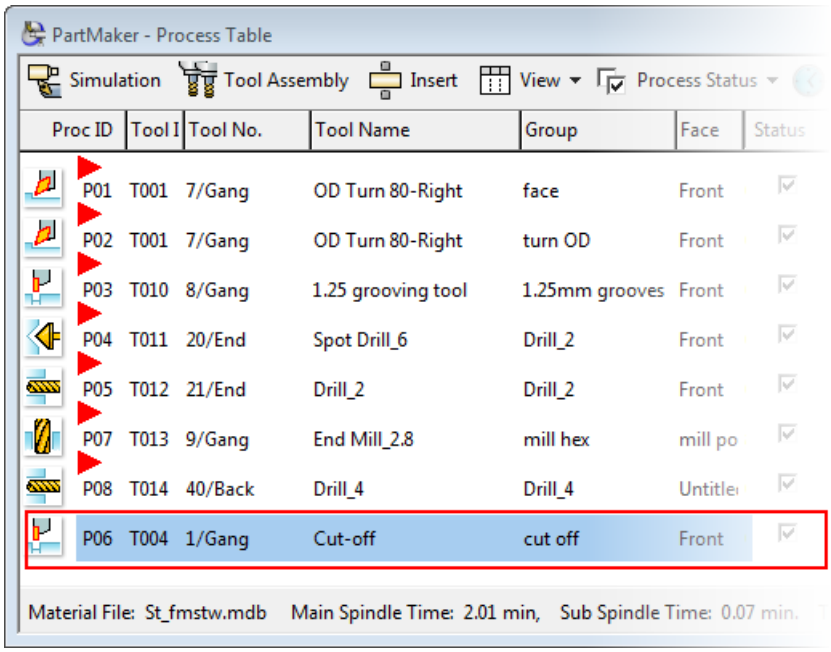
2 Complete the **Process Table Options** dialog as shown, then click **OK**.



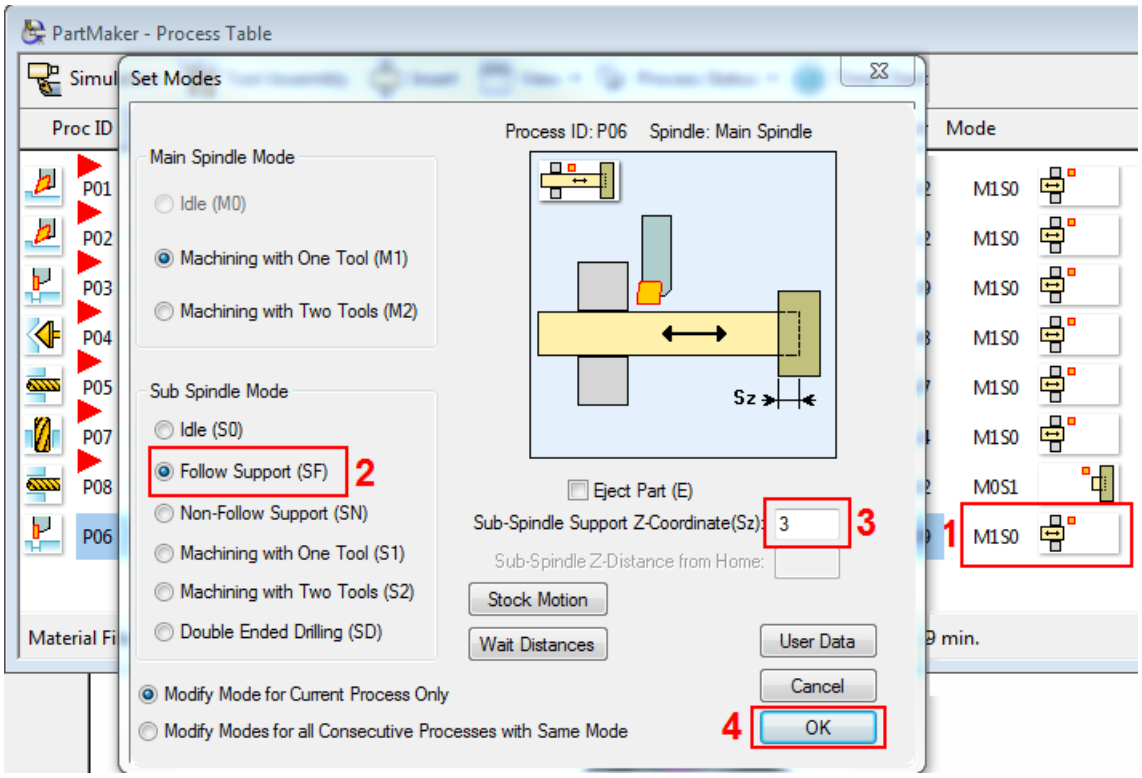
3 When viewing the Process Table, you can see that the cut-off process is not the last process in the table. To move this process to the end of the table, select the cut-off process so it is highlighted, then drag-and-drop it to below the drill_4 process.



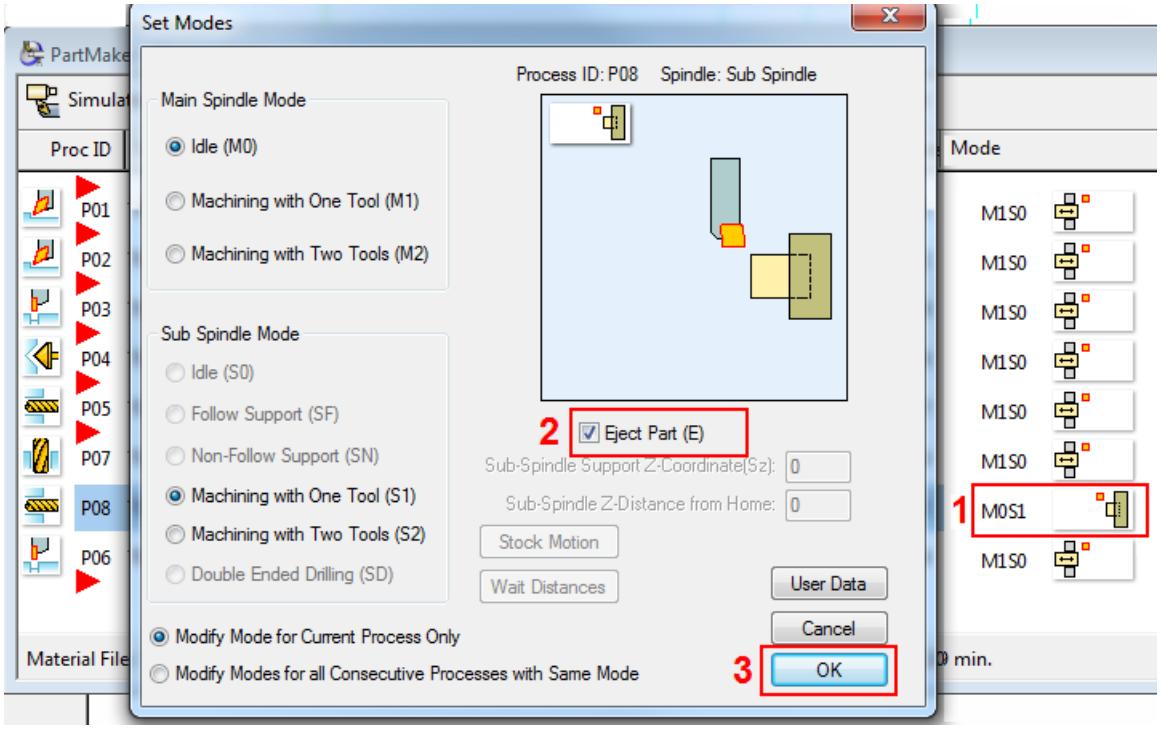
The Process Table now shows the processes in the order in which you want them to be machined:



4 To change the mode of the machine for the cut-off, click the **Mode** button alongside the cut-off process, complete the **Set Modes** dialog as shown, then click **OK**.



5 Add a part eject operation to the last sub-spindle process:

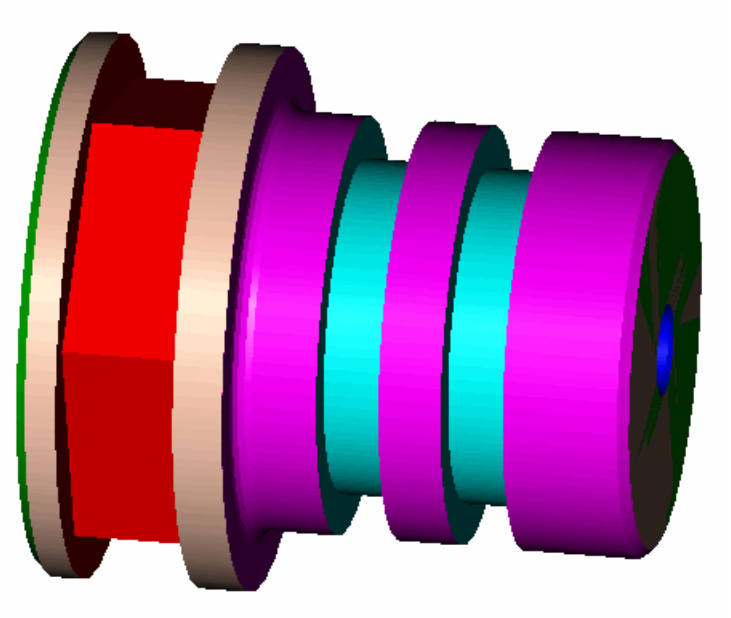


6 To view a 3D simulation of the machining process, click **Simulation** in the Process Table.

7 In the Simulation window, click **Play** to start the simulation.

8 When the simulation is complete, click the **Show Finished Part** button.

PartMaker displays a 3D representation of the finished part:



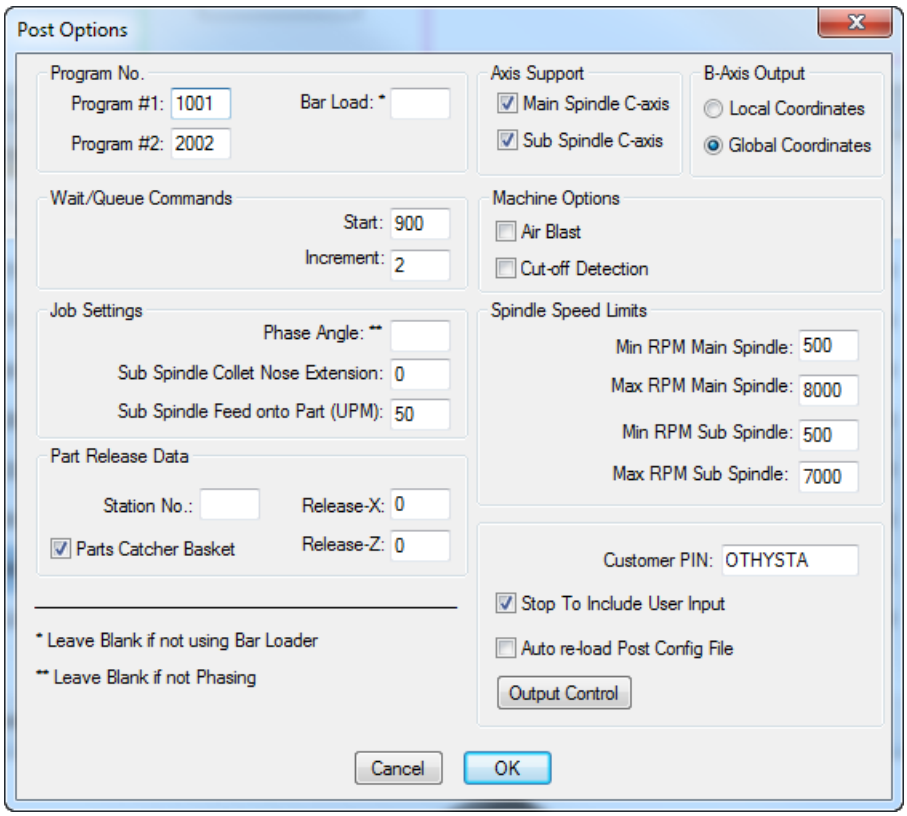
Generating NC Program code

This section describes how to convert the information stored in the Process Table into an NC Program.

This section uses the demonstration post processor (swiss-demo.pst) supplied with PartMaker. This is a generic post processor that is provided for demonstration purposes only. Therefore, the contents of the NC Program file will not reflect the code required for a specific machine and you will not be able to save, or run, the NC Program files that PartMaker has created.

1. Select **Job Optimizer > Post Config File** to display the **Open Post Configuration File** dialog. Select the swiss-demo.pst file in the PM-Swiss folder of your PartMaker installation and click **Open**.
2. Click the **Generate NC Code** button on the Main toolbar, then click **OK** on the **Post Options** dialog to use the default options for postprocessing.

*If PartMaker displays a warning that no ejection is programmed, click **Yes** to continue.*



3 On the **Name NC Program File As** dialog, enter a name for the file where PartMaker saves the NC Program code, then click **Save**.

In this case, because you are using a demonstration post processor, PartMaker will not save the NC Program files.

PartMaker creates demonstration NC program code:

GETTING_STARTED_METRIC.H1

```
%
O1001 (FGETTING_STARTED_METRIC 08/12/2013)
(COLLET RECHUCK)
M200
M94
G97 S1124 T0101 M13
G1 G99 X-0.025 F.002
G0 M21
G4 U0.5
G300 Z15.25
G50 Z0.
M22
G4 U0.5
G0 Z-.015
T0
G0 X2.74
M205
M1

(PROCESS 1 FACE OD TURN 80-RIGHT)
M210
T0
```

GETTING_STARTED_METRIC.H2

```
%
O2002 (FGETTING_STARTED_METRIC 08/12/2013)
(COLLET RECHUCK)
M200
T0
G0 Z8.42
M205
M1

(PROCESS 1 FACE OD TURN 80-RIGHT)
M210
T0
G0 Z8.42

(PROCESS 2 TURN OD)
M215

(PROCESS 3 1.25MM GROOVES 1.25 GROOVING TOOL)
M220
T0
G0 Z8.42
```

Saving your work

This section describes how to save your work and close PartMaker.

1. If you imported the 3D solid model provided for this example, check that the **File > Save Solid Model with Job File** is selected, so the solid model will be saved with the PartMaker .JOB file.
2. Select **Save** on the Main toolbar to save the job file, Tools database file and Cycles database file that are currently open.

3. When the **Save Job File As** dialog is displayed, select the folder where you want PartMaker to save the file, enter a filename in the **File Name** field, and then click **Save**.
4. If you have finished using PartMaker, select **File > Exit** to exit.

Index

T

Z