Desk*Proto*

Tutorial

Including Installation, Quick Start and seven Lessons.

Desktop Prototyping software,
to quickly generate prototypes using a desktop CNC milling machine.

Version 6.1
Copyright © 1995, 2013, Delft Spline Systems.

Delft Spline Systems
PO. Box 2071, 3500 GB Utrecht, The Netherlands.
Internet http://www.deskproto.com

Table of Contents

Disclaimer.....5

Essentials.....7

Installation.....9

Quick Start.....15

Lesson One.....19
The Picture Frame

Lesson Two.....39
The Bottle (two separate halves)

Lesson Three.....53
The Venus Bust (rotation axis)

Lesson Four.....65
The Cell phone (two-sided machining)

Lesson Five.....75
2D Machining

Lesson Six.....85
Bitmap Machining

Lesson Seven.....95
Five-Axis Machining

Index.....107

Disclaimer

All milling devices (whether or not Numerically Controlled) are dangerous devices: when working with a milling machine it is possible to damage either the workpiece or the machine, or even to injure yourself. So do take care, and always check your milling paths before sending them to the machine - in case you are a novice user have an experienced colleague check them.

Delft Spline Systems, the software distributor, the dealer or any other intermediate party are in no way responsible for any damage or injury, direct or consequential, relating to the use of this software.

DeskProto is a registered trademark of Delft Spline Systems.
Windows is a trademark of Microsoft Corporation.
All other trademarks are owned by their respective owners.

Essentials

What does DeskProto offer

DeskProto is a 3D CAM program for 3-axis, 4-axis and 5-axis CNC milling machines, offering **Desk**top **Proto**typing. DeskProto will allow you to machine even the most complex 3D geometry (STL file), any 2D drawing (DXF file), as well as 3D reliefs based on photos (any bitmap file). It can be used for product design, jewelry, woodworking, medical, arts, education, hobby, etc. DeskProto can be combined with any 3D CAD program, and with any CNC milling machine.
Do note that three Editions of DeskProto are available: **Entry**, **Expert** and **Multi-Axis**, the first two offering a subset of DeskProto's functionality.

How does it work

The starting point for DeskProto is an STL file. This file type is standard for all types of Rapid Prototyping, and contains a geometry-description in the form of small triangles connected at their edges to form a surface. Any current 3D CAD-system is able to write this type of file. Also DXF files containing "3D faces" (like from 3D Studio Max) and VRML files can be processed.
It is not possible to create new geometry in DeskProto (it is not CAD software, it is about Rapid Prototyping). DeskProto simply reads a file created by another program and displays its contents. At this point it is possible to scale the geometry, translate, rotate etc. After entering some milling parameters (type of cutting tool, required accuracy, etc) DeskProto will automatically calculate the milling paths. No danger of damaging the model as the paths are gouge-free ! In addition DeskProto can also import 2D files (DXF, AI, EPS) in 2D Operations and bitmap files (BMP, JPG, GIF, PNG, TIF) in Bitmap Operations. Run the toolpath program on the desktop CNC milling machine in your own office, and you will have your model ready: *within a few hours* !

What hardware/software is needed

DeskProto is a MS Windows application, it needs Win XP (SP3), Win Vista, Win7, Win8 or newer. On 64 bits Windows versions a 64 bits DeskProto will be installed, otherwise a 32 bits version. Minimum required hardware is a Pentium PC with 1 GB RAM and 100 GB free disk space: faster/more is better. An OpenGL compatible 3D graphics card is recommended.

Installation

DeskProto version 6 runs with MS Windows XP (SP3), Win Vista, Win7, Win8 or newer. Minimum required hardware is a Pentium PC with 1 GB RAM: faster/more is better. The use of an OpenGL compatible 3D graphics card is recommended. For installation you need about 20 Mb of free disk space, to use DeskProto you need much more for the NC program files that you will create.

You can download the Setup file from www.deskproto.com
Or, in case you have a CD, insert the DeskProto CD in your CD Drive and the Install Menu will be automatically displayed: choose option 1 "Install DeskProto" to start Setup.
After pressing Continue on the security warning the DeskProto Setup will start:

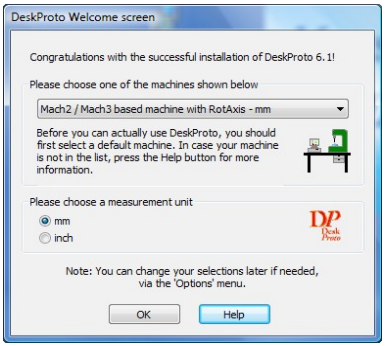


Now just follow the instructions given:

- accept the license agreement
- read the welcome information
- confirm the installation folder
- confirm the start menu folder
- select which icons you want
- and Install.

A new shortcut called DeskProto will be created on the desktop (unless of course you unchecked that option), and the necessary files will automatically be copied to your hard disk. Also the commands DeskProto, DeskProto Help-file and DeskProto Uninstall will be added to the list of Programs that can be accessed via the Windows Start menu.

The first time that you start DeskProto it will complete the installation by asking you which CNC milling machine you will be using and which units (metric or inches).



Select the correct **machine** (the one you have) in the drop-down list: it will be the default machine that DeskProto will use for all your projects. If needed this setting can later be changed in the default Part parameters (Options menu).

If your machine is not listed, in most cases the machine called “ISO plain G-codes”, -inch or -mm, will be your best shot. You may later define your own Machine in the Library of machines (Options menu), for more information see the Reference Manual and/or the Help file.

The **units** that you select here will be used for the geometry file import and for the user-interface. These are the units as set in the Preferences. Keep in mind that units can be set on a second location as well: the units in the NC file are set per machine, in the postprocessor for that machine. Check the Help file for more information about the Units settings in DeskProto.
(the postprocessor is the software module that translates the output for a certain machine, so a sort of Driver).

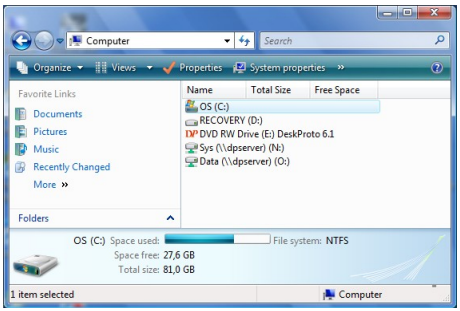
After this Setup you will have DeskProto running on your PC, in Trial mode. This means that for a 30 day trial period you can evaluate if the program indeed suits your needs. When the 30 days have expired you will no longer be able to save NC files.

When you have bought a DeskProto license you will receive an unlocking code that you can use to remove this trial limitation. In the Register dialog (Help menu) you need to enter the Name and the Code that you received. Exactly as specified: with capitals, spaces and comma.

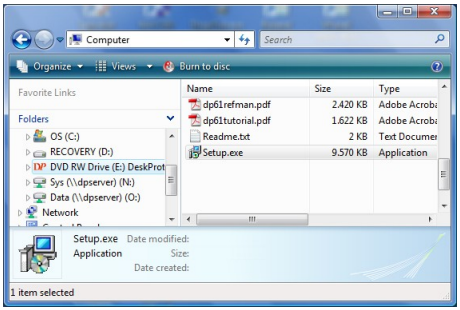
Setup troubleshooting

In case the DeskProto Install Menu does not show up after inserting the CD, you can locate the file Setup.exe on the CD and manually start it. The Setup is a program that can be started just as any other Windows program. You will also need to start Setup.exe after downloading this it from the DeskProto website.

An easy way to Start Setup is via Windows Explorer (not to be confused with Internet Explorer). Start this by double-clicking “My Computer” on your Desktop, or by pressing the Windows key and the “E” key at the same time. Windows Explorer looks like:

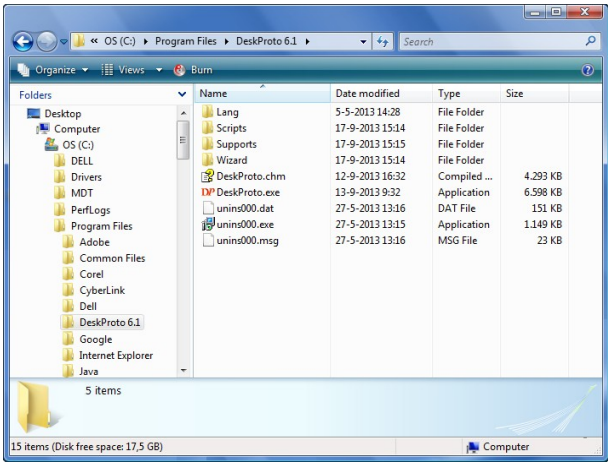


Look for **Setup.exe** on the CD drive (in most cases drive D: or E:), or in case of a downloaded file look on your hard disk (usually drive C:). Do not mind the other files that are shown. You can then start Setup by double clicking on it's line.



Files and directories

DeskProto used to store all files clearly arranged within one directory structure. Unfortunately in newer versions of Windows that is no longer possible, due to the severe security rules by the User Account Control (UAC). Programs, drivers and user data now need to be stored in different file-locations.



By default DeskProto V6.1 stores its files in the these directories (folders):

\Program Files

\Program Files\DeskProto 6.1
This directory contains the executable and the help-file of DeskProto.

\Program Files\DeskProto 6.1\Lang
This directory (empty after an English installation) is reserved for other **Lang**uage versions of DeskProto, and will contain translated files.
Each language needs a resource file in this directory, after which in the DeskProto Preferences the new language can be selected. The file is called *ResourcesLang.dll*, where Lang is the language. For instance, for a German version of DeskProto the file *ResourcesDeutsch.dll* is needed. The second file in this folder is the translated Help file: *DeskProtoDeutsch.chm*
In addition German Cutter-definition files are needed in the Drivers folder

\Program Files\DeskProto 6.1\Scripts
This directory contains all Scripts, see the Reference manual (on your CD and on the DeskProto website) for more information.

\Program Files\DeskProto 6.1\Supports
This directory contains all Supports, see the Reference manual (on your CD and on the DeskProto website) for more information.

\Program Files\DeskProto 6.1\Wizard
This directory contains all files for the Script Wizards, see the Reference manual (on your CD and on the DeskProto website) for more information.

\ProgramData

\ProgramData\DeskProto 6.1\Drivers
For making an NC program that is suitable for your milling machine, DeskProto needs information about the correct machine, the postprocessor and the available cutters. This information is available from the configuration files (*.MCH for the machines, *.PPR for the postprocessor and *.CTR for the cutters), stored in this drivers directory.

\ProgramData\DeskProto 6.1\Samples
For novice users each DeskProto comes with a number of sample projects, also used for the lessons in this Tutorial. This concerns DeskProto Project files (*.DPJ), 3D Geometry files (*.STL which is short for STereoLithography), 2D and 3D files in the Autodesk Data Interchange Format (*.DXF), and a few sample bitmap files.

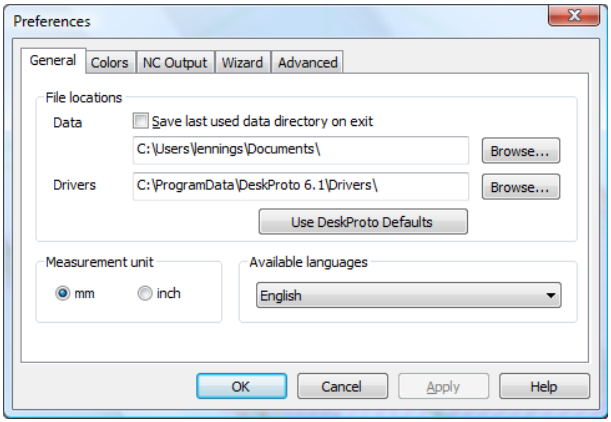
You can easily access the samples via the DeskProto Start screen, by checking the box **Use samples folder**. Easier still is to use the link to the DeskProto Samples at the left side of each File Open and File Close dialog (not present in WinXP).

The location of the folder **ProgramData**\ is different per Windows version.
In Windows XP this folder is called:
C:\Documents and Settings\All Users\Application Data\
For some reason this had been made a hidden directory: to make it visible in File Explorer ('My Computer") open Tools >> Folder Options >> tab View and select the option "Show hidden files and directories").
In Windows Vista and Win7 and Win8 the folder is located in the root (so as in C:\ProgramData\ ).
To make the folder visible here open tab View via Organize >> Folder Options.

My Documents or **Documents**
In WinXP this folder is called My Documents, in Vista, Win7 and Win8 just Documents.
This is the default **Data** directory, so it will be used for all Geometry Load and Save commands and also for the NC program files that you produce. It is

advised to create a directory structure here, for instance with a new directory (folder) for each new client or for each new project.

Some of the file locations are user-configurable:



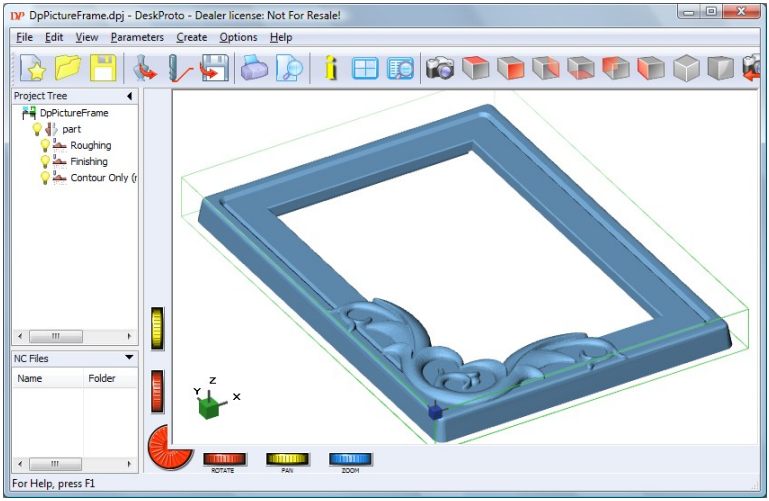
The file locations for Data and for Drivers can be defined in the DeskProto Preferences (Options menu), see the illustration above. The button “Use DeskProto Defaults” can be used to reset these locations.

Following Microsoft's guidelines for Windows applications DeskProto will use the **Registry** to store all program settings, like default parameters and preferences.

Quick Start

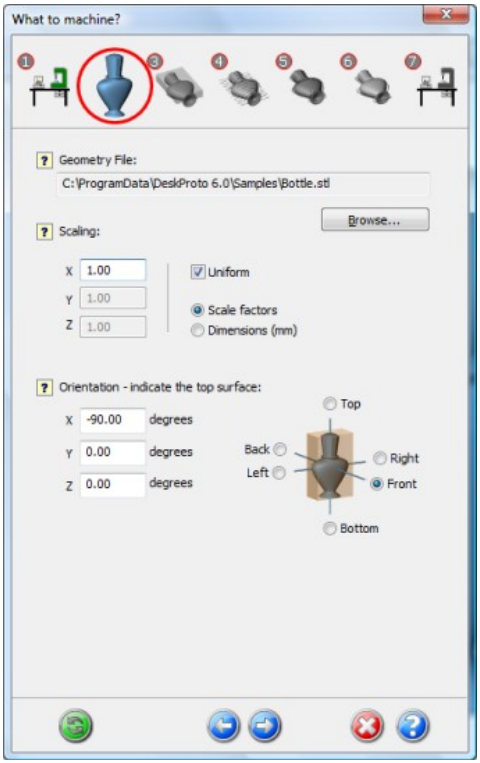
The function of this Tutorial is to step by step introduce you to the functions that DeskProto offers. It is recommended to read **and** execute at least lessons number one and two before starting to make models with your own geometry.

However, if you are not a great manual reader and want to start at once exploring DeskProto, at least read this Quick Start first. It is meant to explain the basic ideas of DeskProto, and you will need this information to be able to understand what is happening.



The DeskProto screen contains standard items like the title bar (top line), menu bar, toolbar (the row of buttons below the menu) and status bar (bottom line). The center area is divided into three tiles: the large View window on the right, and the windows Project Tree and NC files on the left. All these elements will be explained later on in this Tutorial.
You can always use the Help function for extra information on any part of the screen.

For now it is important to know that within this screen two different user-interfaces exist: the **wizard-based** interface and the **dialog-based** interface.



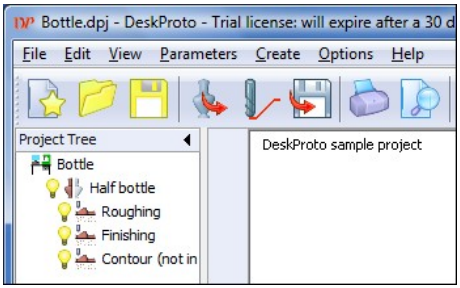
New users are advised to use the **DeskProto Wizards**, that will guide them through all the steps needed to generate an NC toolpath file using their own geometry. The illustration above shows a typical wizard page.

A wizard will set the same parameters as available in the dialogs, only now they are presented in a sequential series of screens, and only the most important parameters are available. You can find the wizards on the Start Screen or via the File menu, for more information see lesson 1A.

When using the **Dialog-based interface** you need to know where to find the parameters. In this interface you can define parameters on three levels:

1. Project parameters include the name of the geometry file and the number of Parts that you want to use to create this prototype.
2. Part parameters define **What will be milled.** They set the size, orientation, position and alike. Within each Part you can use one or more milling Operations.
3. Operation parameters define **How it will be milled.**
These are in fact the only real 'milling parameters'. Three different types of operation are available: in addition to the standard (3D) Operation, a 2D Operation and a Bitmap Operation are available as well.

The **Project** is the central concept of DeskProto. All information about a prototype is stored in a Project file, which is the file to be opened when starting and to be saved when finishing. The project file contains all milling parameters and viewing parameters, and also contains a reference to the **geometry file** (although it does not contain the geometry itself).



You can imagine the tree-like structure of a project, which is displayed in the **Project Tree** at the left side of the DeskProto screen: see the figure above. This sample Project “Bottle” consists of one Part called “Half bottle” and three Operations called “Roughing”, “Finishing” and “Contour”. Each operation line includes a lamp icon that you can switch on and off to make the operation (in)visible. The project will be named when saving it for the first time, until then the tree displays the name “untitled”.

Note 1: three different editions of DeskProto are available: **Entry**, **Expert** and **Multi-axis**. The Entry edition and the Expert edition contain subsets of the available Part and Operation parameters. For the rest the editions are equal.

Note 2: to open a 3D file in DeskProto you must use "**Load Geometry**" in the File menu (if needed start a New project first). You cannot use OPEN, as you do not yet have a DeskProto project file for this new project. The 3D geometry is used for for Parts and for all 3D Operations.

This is different for 2D files and bitmap files, as these are used for one operation only. You can add such special operation in the Part parameters, or after right-clicking on the Part's line in the project tree:
To load a 2D file: create a 2D Operation and edit it's 2D Operation parameters.
To load a bitmap file: create a Bitmap Operation and edit it's parameters.

Any function in DeskProto can be reached using the pull-down menus or using the buttons on the toolbar. The most important menus are described below:

* The **View Menu** offers the opportunity to change the way you look at the geometry. Also try changing your view by rotating the six colored thumb-wheels on the screen, and by using your mouse inside the view window. Most of the functions in the View menu can also be activated by using the button bar.

* In the **Parameters Menu** you can edit all geometry parameters and milling parameters. For simple prototyping it is sufficient to edit only the front Tab screen for both Part and Operation parameters: the other Tabs can come later (all parameters have suitable default values).

* The **Create Menu** is the most important; this is where you can start the milling calculations and write the NC program file.

We do hope you will enjoy using this software, it certainly can help you to make your prototyping really rapid.

Lesson One

The Picture Frame



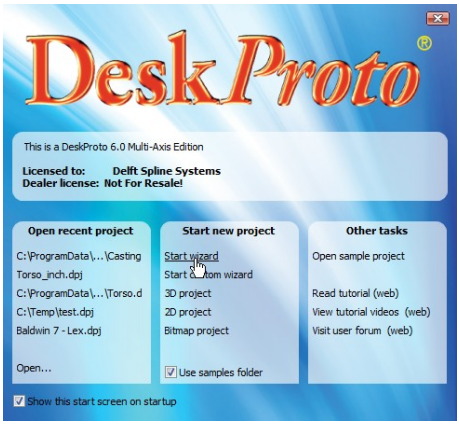
In the first lesson the most elementary functions of DeskProto will be explained: you will learn about the DeskProto user-interface and its main functions. A 3D geometry will be processed and an NC file will be made, ready to send to the milling machine. The lesson will be presented **twice**: first using the **Basic 3D Wizard** and next using the **Dialog-based interface.**
This lesson is for all DeskProto editions.

The geometry is shown in the figure above: a nice **picture frame** with floral decorations. Mill it in wood and add your own favorite picture: a great gift ! This part can be completely machined from one side, making it a a splendid sample model for this first lesson.
The relief has been created by Todd Bailey of 4m3D Creative Design (www.4m3d.com) as a custom model for DeskProto. You can find the file *DpPictureFrame.stl* in the Samples directory that was filled during Setup.

Start DeskProto

You can start DeskProto using its Program Icon on your desktop, using its shortcut in the Windows Start Menu, or in any of the other methods that Windows offers you. When starting, DeskProto will briefly show its Splash screen (the first time an extra dialog will pop up asking you to select the machine and the units to be used, see the Installation paragraph). After that DeskProto will show either the Trial screen or the Start screen.

The **Trial Screen** is shown only when you are running your DeskProto in Trial mode (DeskProto offers a 30 days trial period to try the program before you decide to buy). It tells you how many trial days you have left, it asks you which edition of DeskProto you want to start, and it shows buttons to either order a license or to register one.



The **Start Screen** will make life easier for you as it offers shortcuts to the most common tasks: opening an existing project, starting a new project in various ways, using one of the samples, reading or viewing one of the Tutorials. It is optional though: uncheck the check-box to make DeskProto skip this screen.

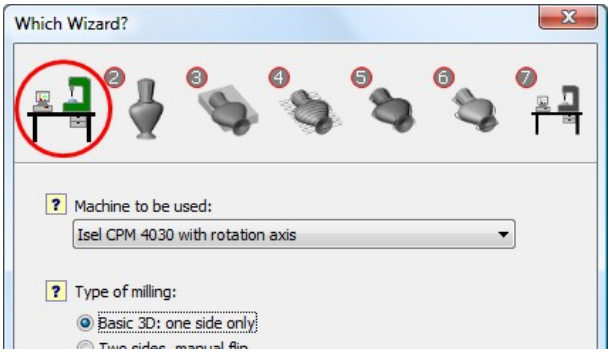
DeskProto offers two user-interfaces: **Wizard-based** and **Dialog-based**.
Important for novice users are the Wizards, as they will step-by-step guide you along all actions needed to create an NC file in DeskProto. All settings offered by the Wizards are also possible in the 'normal' dialog-based user-interface.

In this first Tutorial lesson we will explain both interfaces: the Wizard-based in **Lesson 1A** and the Dialog-based in **Lesson 1B**. So for Lesson 1A now please check **Use samples folder** and then select the option **Start wizard**.

Lesson 1A

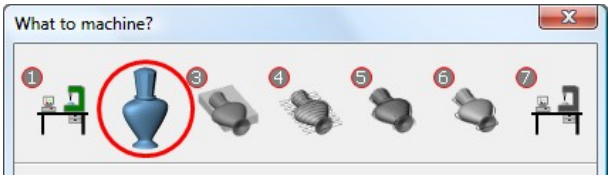
The Picture Frame, wizard interface

You have just selected Start Wizard, either in the DeskProto Start Screen or in the File menu. So you can now use the DeskProto **Wizard interface**, making the program very easy to use for anyone without previous experience. We will keep the Tutorial as short as possible, as the wizard should in fact be self explanatory...



The Machine to be used should already be the correct machine, as you have set the default machine when first starting DeskProto. If not correct you can change the default machine in the Default Part parameters (Options menu).

In fact a series of different wizards is available, each meant for a specific type of milling. For the Picture frame we will use the first wizard: **Basic 3D, one side only**, that is available in all Editions of DeskProto. So please select that wizard and press Next.



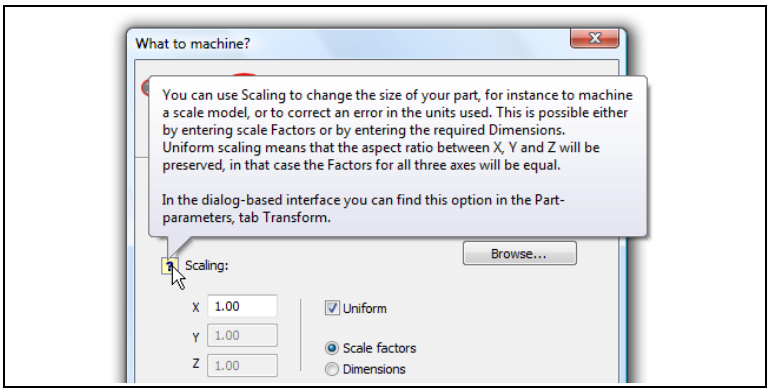
As you can see now the second icon becomes active (enlarged and in a red circle): you are on the second page of this wizard. These icons can be used as navigation tabs: you can click on any of the colored icons to directly go to that

wizard page. Most forward jumps are not yet possible though (gray icons), as you cannot skip any of the wizard's pages. When using the DeskProto **Expert** Edition or **Multi-Axis** Edition you will see **seven pages**, as shown in the illustration. When using the **Entry** Edition you will see **five pages**, as the settings for *Materials and Supports* and for *Contouring* are not available in this edition. Nevertheless also in the Entry edition this lesson can be completed, as we will not apply these extra settings anyway.

On this second page you need to browse a **geometry file**. As you started with option "Use samples folder" checked, the Browse button should directly look in the DeskProto Samples folder: select file *DpPictureFrame.stl* and press Open.
Note for INCH users:
For users that work in inches, most sample geometries are also available in an inch-version. For the picture frame geometry the inch version is called ***DpPictureFrame_inch.stl****. So please select that file, as the metric version will result in a frame of 183 inches high.*

In case you do not see the Samples folder: you can find it in folder \ProgramData\DeskProto 6.1\Samples\ (see the earlier paragraph on Files and Directories)

Scale when you need a different size picture frame, and do not change the Orientation as the Top surface needs to remain on top.
For more information on any of these subjects: move the cursor over one of the **yellow question marks** to get an explanation.



On the third page called "Material and Supports" no changes are needed for this project: use the geometry's bounding box as material block, do not add supports and keep the default zero point position. This page Materials and Supports is not present for users of a DeskProto Entry Edition, as settings for



segmentation, support tabs and translation are not available in that edition.

The next three wizard pages are for the three operations, as this wizard will automatically generate three operations: **Roughing** (optional), **Finishing,** and **Contouring** (optional). The roughing operation is meant to quickly remove material, finishing to create an accurate model with a smooth surface, and the final contouring will smoothen out most staircase steps that may be visible, using the strategy "Contour Only". On these three pages you need to enter the actual Milling parameters like cutter, speeds and precision.
In the Entry Edition only one strategy is available, so then this wizard will skip contouring and create just two operations.

A **Cutter** can be selected from the list in the Combo box (the drop-down list): the list shows all cutters in the Library. You may need to add or change a cutter: then you can enter this Library via the button.
For finishing a freeform surface like this frame a ballnose cutter is the best choice as it will create a smooth surface. The larger the radius, the smoother, the drawback obviously being that for small details a small cutter is needed.
You may select different cutters for each of the three operations, in which case of course a tool change will be needed.

For Roughing select a large **Distance between the toolpaths**, for Finishing a small one. In most cases the default values will be OK for a first model
For the **Speeds** you can also use the default values, unless you are machining in tough material like metal.
The wizard already has selected an optimal **Strategy** for each of these operations. In the Entry edition the strategy cannot be set as only one strategy is available: Parallel.
For the Roughing operation two extra parameters are available: **Skin** and **Layer** height. Leaving a Skin around the model will improve the surface quality: any damages made when roughing (for instance by cutter vibrations) will not be visible in the result. Finishing will then move smoothly as the chip load is constant (only removing the thin skin). The Layer height (how deep may the cutter sink into the material in one go) may of course never be higher than the cutting length of the cutter that is used.

Each of these three Operation pages also shows a field for the **Estimated machining time.** To see the machining time you must press the button Calculate. Then the toolpaths for that operation will be drawn as well.

The last page of the wizard, **Send to machine**, shows the resulting project tree (you can rename any line in the tree after a slow double click).
The button **Send to machine** is available only when your machine supports that option and when an NC output device has been configured in the DeskProto preferences.
The button **Write NC program file** in most cases is the last step to be done in

DeskProto. This file then can be sent to your CNC milling machine, see the end of this Chapter. The file extension depends on the machine that you have selected as your default machine: each machine manufacturer uses a different type of NC file.

You can end the Wizard by pressing Finish. After that you may save this new project via File >> Save: this save will produce a DeskProto project file, with extension DPJ.

However, before you do so please read the below **Notes** about this result:

1. The picture frame geometry has a large hole in the center area. Which makes of course sense for a frame, however which is not ideal for the standard parameters as just set by the Wizard.
The default Roughing strategy is Block and makes the cutter move in from the outside. Perfect for most models, however not optimal for the frame as at some point the remaining material in the center will be cut loose. That loose chunk of material may damage your model, so here it will be safer to use strategy Parallel for Roughing.
The default Finishing strategy is Parallel, which will also finish the empty center area, so will take longer than needed.

2. DeskProto does offer many options to make these toolpaths more efficient, most of these are available only via the Dialog-based interface. For more information see the lessons in the next paragraph.
It is possible to first use the Wizard and then (after finishing the wizard) fine-tune your project using the Dialog-based interface.

3. In the DeskProto Samples directory you can find a sample project file for this geometry, with much better settings. So easiest is to just open this file DpPictureFrame.dpj (or DpPictureFrame_inch.dpj).

From here you can either read the next half of this lesson, about how to use the dialog-based interface, or jump to the paragraph called "To the milling machine" at the end of this Chapter.

Lesson 1B

The Picture Frame, dialog-based interface

Start DeskProto (or restart it), in the Start Screen (see page 20) please again check **Use samples folder** and now select the option **3D project** (in 'Start new project'). We will now show you how to set all parameters in the Dialog-based interface. Any settings that the Wizards make can also be made in this way.

Load the Geometry file

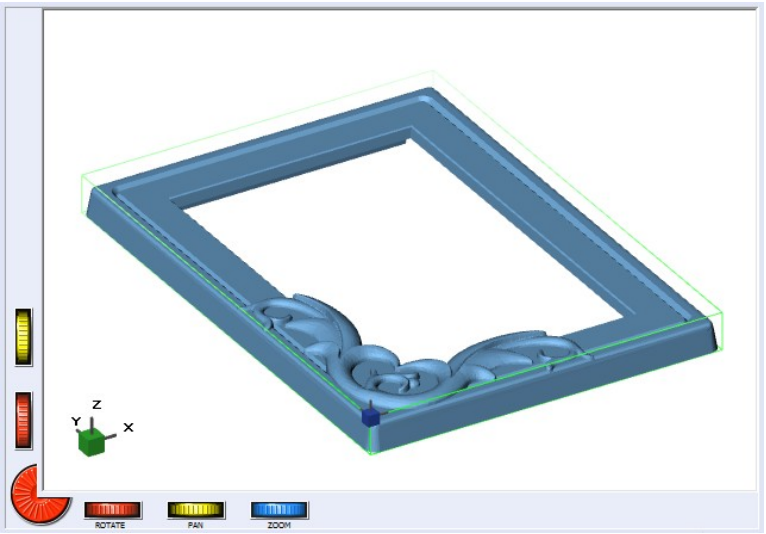
The first thing you need to do for a new project is load the geometry that you want to use. In most cases this will be an STL file. In DeskProto you can do so using the command **Load geometry**, located in the File menu (or using the  Load geometry button). The result will be a File-Open dialog in which you can browse the STL file you want to use. As you selected '3D project' in the Start Screen, DeskProto has opened this File-Open dialog for you automatically.

The file that we want to use for this lesson is called **DpPictureFrame.stl** and is located in the DeskProto Samples folder. Unfortunately the location of this folder – the Windows \ProgramData\ directory – is different per Windows version, so the file may be difficult to find. No problem: you can use the checkbox in the Start Screen mentioned above. In addition a link to the DeskProto Samples folder is visible on the left side of the File-Open dialog (not in WinXP). So please go to the Samples folder and open the file DpPictureFrame.stl

While reading the geometry file a progress indicator will be shown on screen, counting the percentage that has been processed.

Note for INCH users:
*For users that work in inches, most sample geometries are also available in an inch-version. For the picture frame geometry the inch version is called **DpPictureFrame_inch.stl**. So please select that file, as the metric version will result in a frame of 183 inches high.*

The view window



When the geometry file has been read, the View window will show the geometry: a nicely decorated photo frame. The result of this lesson will be a great product, either on your own desk or to be used as a self-made gift !

The green lines around the geometry show the bounding box: the dimensions of the material block that is needed for this part. The **green cube** with X,Y and Z shows the directions of these three axes, and is called the Orientator. The small **blue cube** shows the position of the WorkPiece zero point (0,0,0) for this part, more about that will follow later.

Rotate, Pan and Zoom

DeskProto offers several ways of rotating and moving the object on the screen, so of viewing the geometry from any side. The controls that attract most attention are the colored thumb-wheels in the border of the view window.

The vertical and horizontal red thumb-wheels offer rotation, around a horizontal and a vertical axis (horizontal and vertical on your display screen). The three-quarter-round wheel in the corner allows you to rotate around the axis perpendicular to the screen. Position the cursor over one of these red thumb-wheels, press the left mouse button and keep it pressed while moving the mouse. You will see that the geometry rotates as indicated.
The two yellow thumb-wheels can be used for panning (moving the geometry

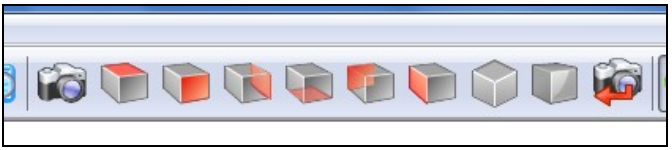
over the screen, horizontally and vertically). And the blue thumb-wheel is meant for zooming (changing the viewing-distance).

While these thumb-wheels attract most attention, in fact they are not needed as you can also use direct mouse control to rotate, pan and zoom. Most intuitive is the Mouse Rotation: position the cursor inside the drawing area, press the left mouse button and move the mouse. The geometry will now appear to rotate. Imagine a large hollow glass sphere around the geometry: the cursor grabs the sphere and rotates it including its contents.



The result of the mouse movement just described depends on the status of the **Mouse function buttons**: see the illustration above. Of these four buttons always exactly one is active (depressed), enabling either mouse rotation, mouse panning, mouse zooming or zoom window. No further explanation is needed: just press one of the four buttons and see what happens when you use the mouse inside the View window.

It is even possible to rotate, pan and zoom without using these mouse function buttons: press the middle mouse button (the mouse-wheel) for panning, and rotate this wheel for zooming. When zooming the cursor position sets the center of the zoom, so you can zoom onto any detail on the screen.

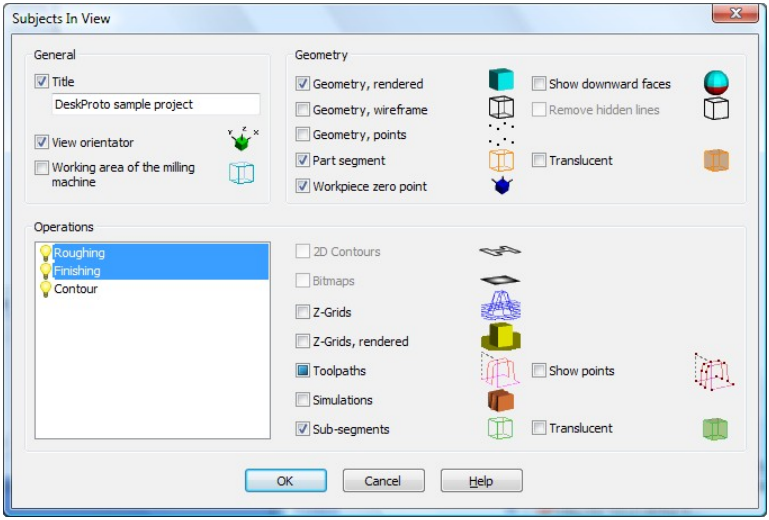


A number of standard views can be set very quickly using the six toolbar buttons showing small cubes. Each button sets a **main view** (a view along one of the main axes). The next three buttons to the right of these cubes can be used to quickly set isometric view, default view and previous view.

Note: all these controls only influence your view of the model (the camera position), not the actual orientation of your geometry on the milling machine.

Subjects in View

An option that will prove to be very handy allows you to select which subjects should be displayed in the View Window. You can open the dialog Subjects in View by clicking Subjects... in the View menu or by double-clicking inside the view window (more methods are possible).



Every displayable subject is shown, with a checkbox to mark each subject to be displayed. When checked (“V”) the corresponding subject will be displayed after pressing the OK button.
When the checkbox is completely filled (showing a square instead of a V), like in the illustration above for the Toolpaths, it means that multiple operations are selected, with a different status for that subject.

For now please look at the geometry subjects only, and play around with the available options to see what happens. For instance 'Show downward faces' is a great tool to find any undercuts in your geometry (area's that the cutter cannot reach), and the option 'Translucent” for the Part segment (the material block) will make it perfectly clear whether or not the part fits inside the block.

Check the model's orientation and dimensions

While examining the geometry by looking from some different directions, you may have noticed that the Picture Frame is correctly positioned for milling. In DeskProto the milling tool always comes from the positive Z-direction: X is from left to right on the milling machine, Y is from front to back, and Z is up and down. As the picture frame is lying flat on it's back, the freeform front side can be machined completely.

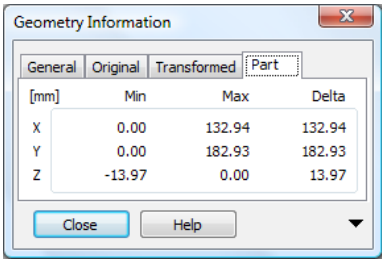
Two features to check the orientation have already been discussed earlier:
- the Orientator (green cube with three axes) shows the direction of the axes
- the option Show downward faces can be used to check for undercuts.

This geometry does not need to be rotated, for a geometry where rotation is needed see Lesson 2.

The above is not completely correct: on the back side of the Picture Frame geometry a recess has been modeled for the photo and the glass, and this recess cannot be created when machining only the front side. Still the frame also can be used without the recess, so in this lesson we will simply ignore it. Information about machining from two sides will follow later.

What you have not yet seen are the dimensions of the geometry, which will tell you whether or not your part will fit on your machine. DeskProto will of course warn you if it is too large, however you will anyway need to know this 'Geometry information' to prepare the block of material you have to use.





Press the Toolbar button with the yellow **i** or select Geometry Info from the View Menu in order to display the Geometry Information dialog: see the pictures above. This dialog gives you the dimensions of the original geometry as present in the file (Tab 'Original') and of the geometry of the model that you are creating (Tab 'Part'). More about the differences between these tab pages will be given in one of the next lessons.

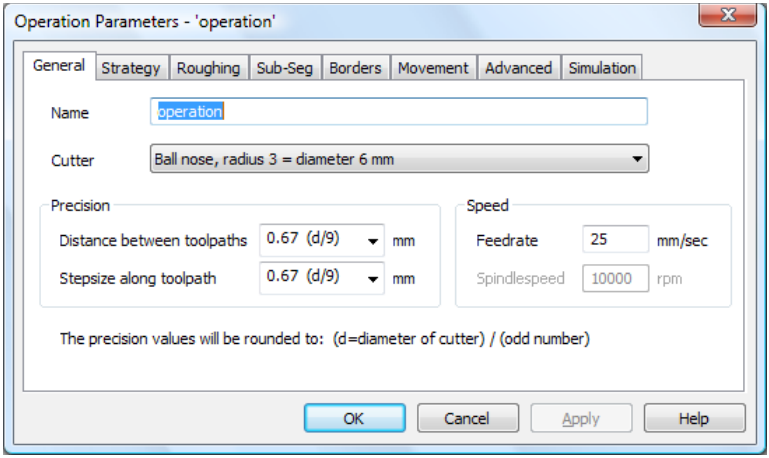
As you can see the dimensions of the part are OK (the illustration is in mm), so you can proceed and start creating your first NC file.

Set the milling parameters

In order to calculate the correct NC toolpath (the path that the cutting tool will follow during the cutting process) DeskProto needs information about the milling parameters that you want to use. For instance the diameter of the cutter to be used, and how accurate you want to have your model. In this lesson we will first show these basic parameters, and next teach you how to refine the milling process by using Roughing and Finishing operations.

Obviously it is also important to select the correct machine for your project. We will assume that you already selected your machine when first starting DeskProto: then your machine is the default one so we do not need to select it for each new project.

Cutter and accuracy need to be set for every milling operation. The cutter that you use on the machine must of course match the cutter that you selected in DeskProto: machining with any other cutter will result in an incorrect model. You can find both the Cutter and the Precision in the Operation Parameters:



You can open the Operation Parameters dialog via the Parameters menu, it is easier though to just **double-click** the line of this operation in the Project Tree. This project tree is visible on the left side of the DeskProto screen, and shows all parts and operations in this project (as just explained in the Quick start section of this Tutorial).

The Operation Parameters dialog consists of a number of Tab screens. As all milling parameters have suitable default values, and as we want to start simple: for now only look at the front Tab and just ignore the hidden Tab screens. Do not ignore the most important button in the dialog: the **Help** button! It will lead

to a page that completely explains this one dialog. Please try the Help, and remember this for when you have any question later.

As you can see, a **Cutter** with a ballnose tip of 6 mm diameter has been chosen, which is the default tool in DeskProto (for inch users diameter 1/4"). For freeform surfaces a ballnose cutter is the best choice as it will create a smooth surface. The larger the radius, the smoother, the drawback obviously being that for small details a small cutter is needed. Because of the small details in this geometry you may select a smaller cutter, still this 6 mm ballnose will produce a good result as well (DeskProto will not damage the geometry when the cutter is too big, just leaving rest-material where it cannot reach).

Two Precision values can be set: the meaning of **Distance between toolpaths** will be clear, the **Stepsize along toolpath** will need some explaining. Each toolpath consist of a large series of small linear movements (in CNC terminology: G1 movements). This second setting determines the size of these linear movements (steps). In most cases it is best to set equal values for both Precision parameters.

Smaller precision values will lead to a smoother and more accurate result, however also to a longer machining time. You can ask DeskProto to estimate the machining time in the Create menu (see the next paragraph).

For machining foam, tooling board and wood the default values for **Feedrate** and **Spindle speed** will be OK; generally speaking these need to be changed for harder materials only. So now press OK to close the Operation Parameters.

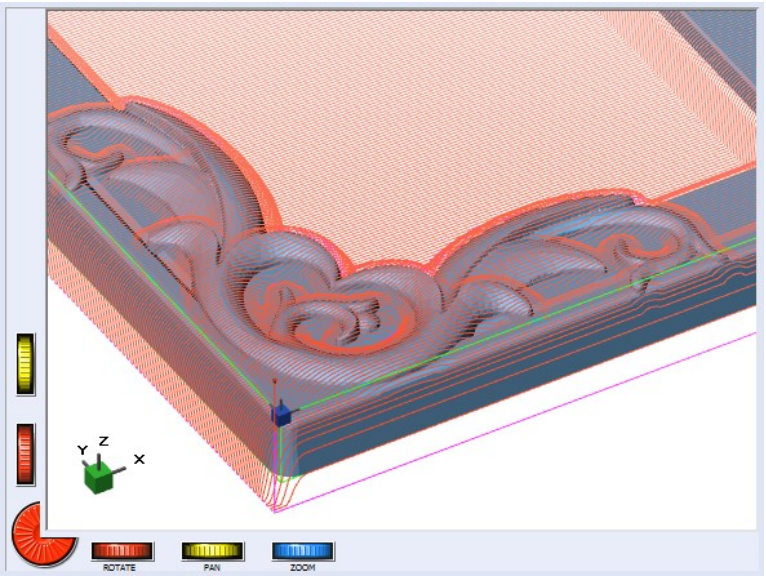
Calculate the Toolpaths

After having set the milling parameters you can now calculate the toolpaths: in the Create menu select Calculate Toolpaths, or (easier) press the button



"Calculate toolpaths" (5th button).
During the calculations DeskProto shows a progress-bar to keep you informed about their progress.

After the calculations have finished DeskProto will display the toolpaths: the red line is the path that will be followed by the tip of the cutting tool. First and last point of the toolpath are indicated by small red arrows. Some of the toolpaths may be drawn using dashed gray lines: these are the positioning movements of the cutter above the model (at Free movement height), which will be done faster (called Rapid) than the cutting movements (done using the Feedrate). Also the rising to this "Zfree level" after the last cutting movement is done in Rapid mode.



The above picture was made after zooming in a bit, in order to actually see the toolpaths in the illustration. You can clearly see the distance between the toolpath and the actual geometry: this is the 3D compensation for the Cutter radius that DeskProto has calculated.

Create NC program

To send the toolpaths just calculated to your milling machine you will have to first save them in a file, called the NC program file. You can do this in the Create menu, with command **Write NC program file**, or (easier) by pressing the button "Write NC-file":


Load geometry


Calculate toolpaths


Write NC-file

You will have seen that these three buttons (no 4, 5 and 6) are the central buttons in the DeskProto work flow.

After giving this command a 'Save-as' dialog box will appear in which you can enter the name of the NC program file to be written. The file extension depends on the machine that you have selected as your default machine: each machine manufacturer uses a different type of NC file. After pressing the Save button DeskProto will write the NC program file to disk. As all calculations have already been done, the process of creating an NC file will not take much time.

Note 1:
For some machines it is not needed to write an NC file, as DeskProto can directly send the toolpaths to the machine. This can be done using command Create menu >> Extra >> Send current toolpaths to machine.
This option must first be configured, via Options >> Preferences >> Tab NC output >> select and configure the NC Output device.
Not many machines support this: we know that all machines by Roland do so, and some high-end industrial machines.

Note 2:
What you just have done is write the toolpaths. Do not confuse this with saving the project, which is the standard Windows File >> Save action and writes an **DeskProto project file**. In the project file all parameter settings are stored, and a link to the geometry file that is used. DeskProto project files have the extension DPJ.

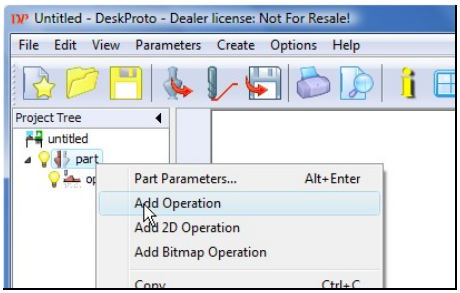
From here you can either proceed to paragraph "To the milling machine", or first learn about roughing and finishing in the next paragraph.

Roughing and Finishing

The toolpaths just made will indeed produce a picture frame, so you are free to skip the fine-tuning as presented in this paragraph. Still, applying Roughing and Finishing has a few distinct advantages, so it will be worthwhile to bear with us for some more time.

You want the machining time for your part to be as low as possible, and you also want an accurate model with a smooth surface. When using only one operation you need to choose between quick or accurate though. As the cutter cannot remove all material in one go (except for very thin models) it will move down in layers. For instance sink 5 mm into the material, remove all excess material above that level, next sink to minus 10, and so on. DeskProto will automatically apply this layering (which is a roughing functionality) as it does not allow the cutter to sink deeper into the material than its cutting length (at least: not in the first operation). When doing so with a small Path distance (needed for a smooth surface) this will take a long time.

When using Roughing and Finishing:
the **Roughing Operation will quickly remove material** (using a large toolpath distance), after which
the **Finishing operation will produce an accurate model** and a smooth surface (using a small distance).



In order to achieve this we need to have two operations in DeskProto, so you need to create a second operation for the current part. You can do so in several ways: shown above is a right-click in the Tree on the line of the Part, and then in the Context menu select Add Operation. The result will be a tree with two operation lines, called “Operation” and “Operation [#1]”.

Double-click on the first operation line in the tree and change its name to “Roughing”. This concerns the operation that you have opened in the previous paragraph: in case you have made many changes you can better first switch Operations 1 and 2 using 'Move up' or 'Move down' in the context menu.



Now you can set the Operation parameters to make this a real **Roughing Operation**. Go to the Roughing tab and select a Layer height (the default value is the Cutter length which will be too much in many cases) and the Skin thickness. For a 6 mm ballnose cutter and a soft wood you could use say 10 mm Layer and 0.5 mm Skin (in inches: for a 1/4" cutter this is 0.4"Layer and 0.02"Skin). You can also check the Ramping option. Use the Help for more information about these settings.

As Roughing Strategy (2nd tab) we often use Block as in most cases this is most efficient. Not for this frame though, as this will result in a loose block of material in the center, causing trouble. So leave the strategy on Parallel.

On the General Tab you can now select larger Precision values (Distance and Stepsize). In most cases the second value in the drop-down list will do: d/3. This means 1/3 of the Cutter diameter, so you'd expect 2 mm (0.0833"). Instead d/3 now will say 2.33 mm (0.0967"). Reason for this difference is the Skin that was just applied. The skin is processed by calculating with a 'virtual cutter' that is thicker than the real one: R 3 + skin 0.5 = R 3.5. This means a diameter of 7, and 7.0/3.0 results in 2.33.
Close the dialog using OK.

The second operation will be the **Finishing Operation**: open the parameters, and change the name to 'Finishing'. You do not want any Roughing functions active, and you want smaller Precision values (Distance and Stepsize). For finishing we often check the option 'Skip horiz. Ambient' (Advanced tab, also uncheck "ignore enclosed ambient"), which for the frame will skip the center hole. It will result in a shorter toolpath only when on the Movement tab the option Sort has been checked. You can leave all others as they were (default values).

You can choose to use two different cutters for Roughing and for Finishing: a thick flat cutter for quick and efficient roughing, and a small ballnose cutter for detailed finishing. As this geometry contains many small details this will produce a very good result. On the other hand: the alternative of using the same cutter for both operations has the advantage that no tool change is needed.

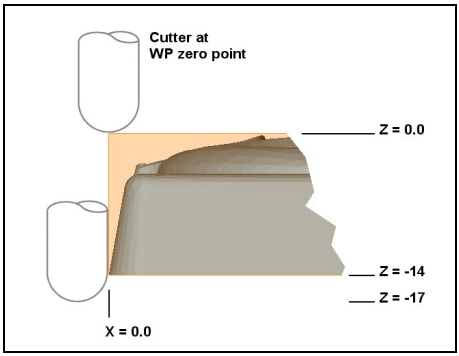
To the milling machine

The created NC program is ready to be sent to the milling machine, so you are finally ready to start cutting off chips now. As the way to do this depends on which milling machine you use, not all necessary information can be given here: please consult the manuals of your NC milling machine as well.

First a block of material has to be prepared. You already know the dimensions of the frame as you have just checked these in the Geometry Information dialog: 133 x 183 x 14 mm, or in inches 5.23" x 7.2" x 0.55.

For **a first test model** you can use a much larger block of material, leaving excess material on all sides to clamp the block without the risk of damaging your clamps. Make the block at least 10 mm larger in the Z-direction, as the ballnose cutter will go R mm lower than the bottom of the frame (R being the Radius of the cutter). This is needed in order to completely machine any vertical and steep surfaces, see the illustration below for a ballnose cutter with R= 3 mm)

You can fixture the block on the machine using clamps, a machine vise or any other method. For light materials like PolyURethane (PUR) or PolyStyrene (PS) foam you can use double-sided adhesive tape.



Next you will have to tell your machine where to find the block of material. In other words: you have to enter the **WorkPiece Zero point** for this NC program, relative to the block just fixtured. A CNC milling machine typically has two zero points: the machine zero point, in a corner of the machining area, and a workpiece zero point (WP zero, also called Program Zero) to be freely defined. As a result also two different coordinate systems are present: machine coordinates (used to define the workpiece zero point), and workpiece coordinates (used for all milling operations).



By default DeskProto sets the **Left-Front-Top corner** of the material block to be (0,0,0). This is the default translation setting. All X and Y positions of the part then are positive (X=0 is the left side of the block, and Y=0 is the front side), all Z-positions are negative (Z=0 is the top of the material block). So the left-front-top corner of the block should be set as the workpiece zero point on the machine. In most cases also will be the starting point of the toolpath.

Or, if your block is larger the zero may also be located inside the block, leaving sufficient room (133 mm for X and 183 mm for Y) to complete the whole part. Or in fact even a bit more as on all four sides the cutter has to move outside the part to machine the outer surfaces.
Make sure to double check X and Y: if you mount the block with the longest side in the wrong direction the part will not fit inside the block.

On many machines you can enter the WP zero point by manually positioning the cutter (milling tool) exactly on the desired workpiece zero point, and then telling the machine controller that this is position (0,0,0). Keep in mind: for X and Y the center of the tool must be positioned, for Z the tip of the tool. Of course it is necessary first to mount the correct tool in the machine's spindle, as different cutters will have different lengths.

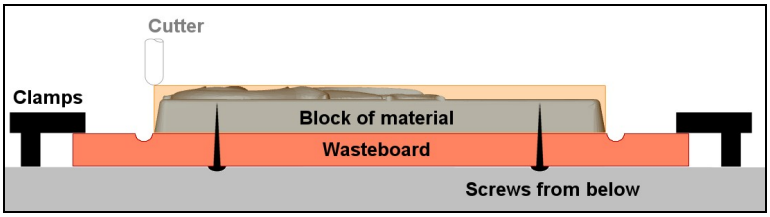
Now you are ready to start the machine by sending the NC program file you just created to the machine. Most CNC milling machines have their own **control software** to do this (like Mach3, PCNC or LinuxCNC). If so then exit DeskProto, start this machine-control program and open the NC program file. If needed first transfer this file from the DeskProto PC to the machine control PC.
Other machines (for instance many Roland machines) can be simply started like a printer. With these machines it is possible to send the file directly from DeskProto by choosing the option 'Send NC Program to Machine...' in the Create menu. In this last case: make sure that the correct communications port or printer driver has been configured (choose 'Preferences' in the Options menu).

At the end of the milling process the model will still be attached to the remaining block of material, as for this first test your block was larger than the model and the three axes milling machine cannot machine the bottom of the prototype. You can either leave it that way (in case you already can see all details that you need), or remove the block using for instance a small band saw machine.

For **the definitive model**, so a frame without any excess material, the block needs to be more accurate and the fixturing more precise. Make the block some millimeters larger for both X and Y, to compensate for possible positioning

errors (not too large as then the chips cannot easily fall off during machining). Make the Z (thickness of the block) as exact as you can. Setting the WP zero point now needs to be done accurately, exactly at the Top Front Left corner of the block. This will be easy, as the process is the same as for the test just done.

Only the fixturing will be different now, as now you cannot clamp the excess material around the model (no excess material is present). So the frame needs to be fixtured from below. This can be done either via double sides tape, or (better) using a few screws from below.



The screws method works out nicely: see the illustration above. You can safely use clamps to securely fixture the wasteboard as the cutter will not come near these clamps. You only need to take care to correctly position the screws: they may not connect to a part of the block that will be milled off !! The waste-board will be 'wasted' as the ballnose cutter will machine a groove all around the model. Also see the **Tutorial videos** on the DeskProto website for a demonstration.

Obviously more fixturing methods are available for this job.
You can for instance also use support blocks (support tabs, bridges) to keep the picture frame connected to the machine during milling. And you can machine the frame from two sides, in order to also machine the cavity on the back side of the frame. More about such advanced options will follow in the later lessons.

If the frame does not fit inside your machine as it is too large: in DeskProto it is easy to scale the model in order to make it smaller. You can do so in the Part parameters. Scaling of course also can be used to make the frame OK for a smaller or a larger picture.
In some case a rotation round the Z-axis of 90 degrees may help, in case the longest dimension of the frame fits your working area along the X-axis but not along Y.

Lesson Two

The Bottle (two separate halves)



In this second lesson you will make a more thorough acquaintance with DeskProto. This geometry cannot be machined from one side, so the bottle will be milled in two separate halves, to be combined to make a complete model (alternative methods are rotary machining and two-sided machining, see the next lessons). This lesson is for all DeskProto Editions, and will use the Dialog-based interface.

The geometry was modeled in a CAD package called SIPSURF (no longer available), by Iris Timmers, at that time an industrial design student. Only the outside geometry has been modeled: it is a massive (solid) bottle. It has been exported as a STL file, and the separate cap of the bottle is included in the same file.

Start a New Project

As any DeskProto session starts with a brand new project you do not need to explicitly start a 'new' project. This is only necessary if you have been working on another project previously: then you have to close that project and create a new using the command New in the file menu, or the New button.

Now use **Load Geometry** (File menu) or use the Load Geometry button to load the geometry file for the Bottle. It is located in the DeskProto Samples folder: just as in Lesson 1 use the checkbox in the Start Screen or the link on the left side of the File Open dialog to find that location.

Select the file BOTTLE.STL that has come with DeskProto as an example (inch users choose Bottle_inch.stl). Note that Windows may hide the extension '.STL' and call the file a "*Certificate Trust List*". You can ignore that false information and just open the file.

In the project file (that can be saved later) a reference to this geometry file BOTTLE.STL will be included, through which this file will be found and loaded automatically the next time the project is opened.

Checking the geometry

The first thing to do after loading the geometry file is to check the geometry: make sure you indeed see a perfume bottle, and check if its orientation and its size are correct.

An easy way to check the **orientation** is to use a predefined Views Layout: View menu >> Layout... In this dialog select one of the options on the right side: T/F/R/Def, in order to see four views: Top, Front, Right and default (3D). You can see that the bottle is standing 'upright': its largest dimension is along the Z-axis. Since the cutting tool will come from the positive Z direction, the model cannot be machined as it is now. You need to change the orientation (rotate), and in a moment we will show you how.

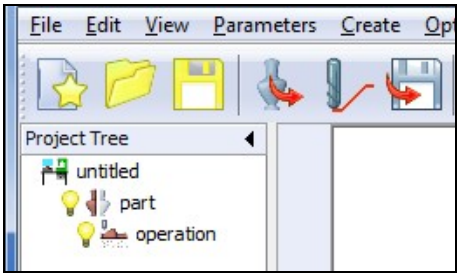
The **dimensions** can be checked using the button Geometry Information: the button on the DeskProto toolbar with the yellow [I], tab Part. For this file the dimensions should show a model of 54 x 29 x 86 mm: a nice perfume bottle, which (after rotating) will fit the working area of your milling machine. For inch users the dimensions should be OK as well (2.12 x 1.14 x 3.4 inch)

Now reset the Views layout to one view only. The Geometry information dialog may remain open when working with DeskProto.

Edit Project parameters

What has to be done now is entering the parameters. When opening a new project DeskProto already gave default values to all parameters, however some of them will have to be changed for this specific model.

In the Parameters Menu you will see that three groups of parameters are present: a group called 'Project parameters', one called 'Part parameters' and another one called 'Operation parameters'. Only one Project is present, in which a number of Parts can be defined. Each Part in turn can contain one or more Operations. This is shown in a tree-like structure, as is clearly visible on the left part of your screen. The standard Windows name for this figure is in fact the **Project Tree**.



In the Parameters menu select Edit project parameters. Or, much easier: simply double-click the project line (the topmost line) in the tree.
The Edit Project Parameters dialog box that you will see now does not contain many parameters. Most important are the name of the geometry file and the names of all Parts: a Part is machined in one fixation of the material block.

In case you would have needed more than one part you could have added new parts here. For now just ignore the other parameters in this dialog box, and also its second tab page. So no action has to be taken here: you can leave the dialog box using the Cancel button.

For many projects (like in lesson one) one part is sufficient. For more complex models more than one part needs to be milled: like for an electric drill you will separately mill a right side and left side, to be glued together later. Then in DeskProto two parts are needed as well, one for each side. For this bottle however both parts are equal, so for the software one part is sufficient.

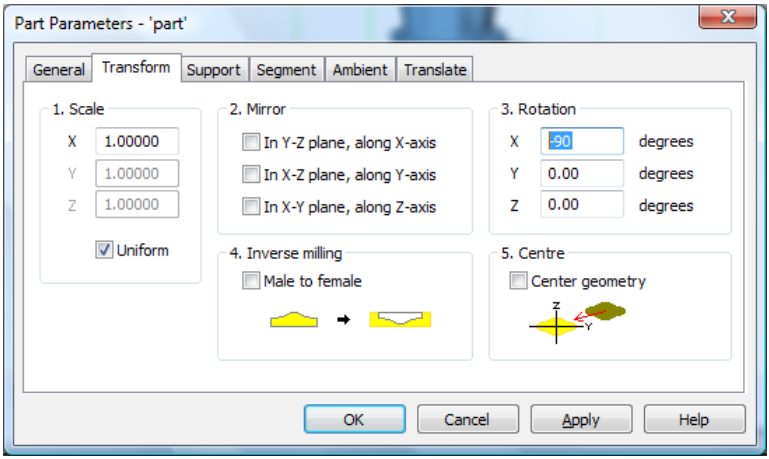
As you can see your new project does not yet have a real name (it is called 'untitled'). You cannot enter a name in the project parameters dialog: it will be asked when you save the project for the first time.

Edit Part Parameters.

Each part has its own set of parameters, to be set in the Part Parameters dialog. Just as with the project parameters it is easiest to open this dialog by double clicking its line in the Tree.

The Part Parameters are presented using a number of Tab pages.
Part Parameters define the geometry to be milled.
Earlier in this lesson, when viewing the geometry, we already concluded that the geometry was not correctly orientated. You will do so now: the geometry rotation is one of the part parameters.

The first Tab page (General) does not need any changes for this project. The only parameter you have to check is the Machine: is the machine that is shown indeed the machine that you are going to use ? (it should be, as you selected the correct default machine during the installation of DeskProto). If not, change the machine using the button at the right side of the name (combo-box button). If you like you can also change the name of the part, however that does not effect the prototype. The number of operations does not need to be changed: one operation is correct here.

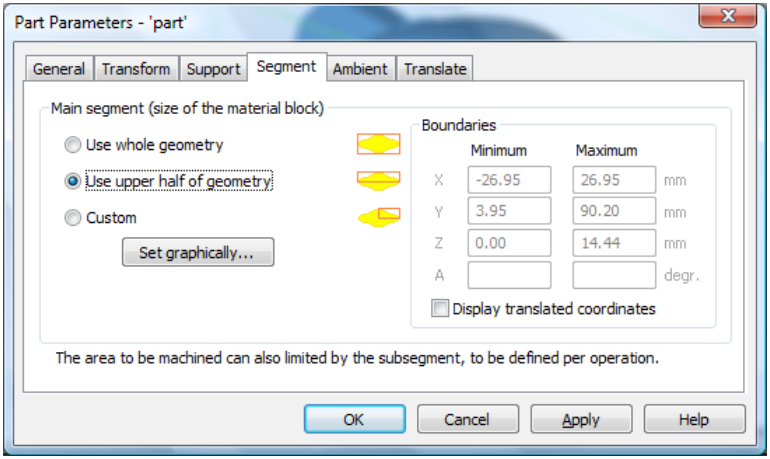


For this bottle model we need to set parameters both on the second tab (Transform) and on the fourth (Segment). The **Transform Tab** shown above makes it possible to change the size and orientation of the geometry. For this bottle (as said) the orientation is not correct: enter a rotation of -90 degrees around the X-axis, and see what happens after pressing Apply. The orientation of the geometry should be correct now. When the part is too large for your machine a rotation of 90 degrees round the Z-axis may be useful as well (for most milling machines the X-axis is the longest).

Using the Apply button is not needed: pressing OK also causes an implicit Apply. Still it may be handy in order to see what happens: if this is not what you need you can enter a different number without needing to reopen the dialog. This also work for scaling: on Apply the dimension shown in the Geometry information dialog (tabs Transform and Part) will be updated.

The other parameters on the Transform tab are OK and will not be covered in this lessen. For more information on these parameters see the Help pages and/or the Reference manual.

You should clearly understand the difference between rotating the geometry and rotating the view:
Rotating the geometry will change the model that is created. You can see on the screen that the geometry rotates while the XYZ axes-cube (the orientator) remains the same. The milling tool comes from the positive Z-axis direction, so because of the rotation a different side of the geometry will be milled.
Rotating the View does not affect the model, it only changes the picture on your screen (the camera position). You can see on the screen that both the geometry and the orientator rotate identically, so the position of the geometry relative to the milling machine remains unchanged.



The fourth Part Tab is Segment, and for this project Segmentation is needed. You now see the geometry of the complete bottle. However, this complete geometry cannot be milled in one part using a three axis milling machine as the cutter cannot reach the bottom half. As stated the model will be milled in two separate halves, so we now want to calculate toolpaths for half a bottle.

Segmentation means to use only a part of the geometry, being defined by a

rectangular block: the segment, which in fact is the material block to be used. By default the segment includes all geometry (default segment is the 'Bounding Box'). We want to machine half a bottle, and in fact exactly want to halve the bottle in Z. This is easy as it is a predefined option: select option 2: “Use upper half”. You can see that at that moment the Minimum Z will change from -14.44 into 0.00. Press OK to exit this dialog.

Notes on the segment:
1. In the DeskProto Entry edition segments are not possible: instead use the parameter “Bottom level” on the tab page Transform.

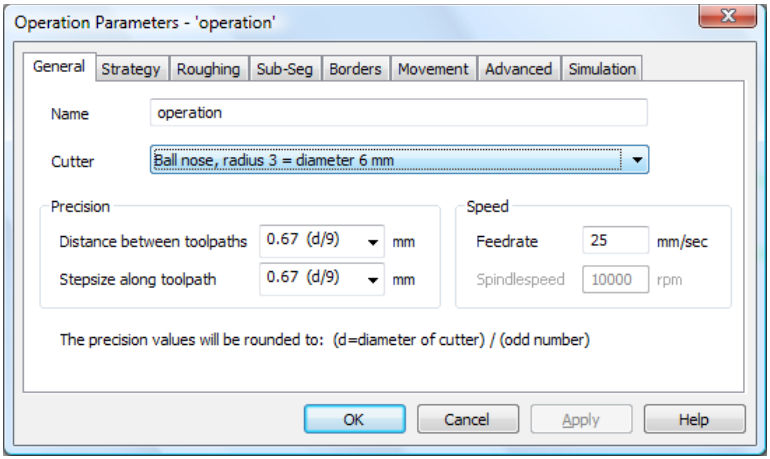
2. In the drawing the part segment is drawn in orange lines, however as after that the operation segment (which is identical) is drawn in green, all you can see is a green rectangular block.

3. All values for the segmentation are given in the transformed coordinate system (so after the changes as defined on the Transform tab), not in the original geometry coordinates. In some cases it will prove to be easier to use the coordinates as used on the milling machine, so to enter the limits in translated values (after the changes as defined on the Translate tab). You can do so by checking the box “Display translated coordinates”.

4. When you select option Custom for the segment, you can define any rectangular block by entering X, Y and Z Boundary values. Still easier is setting a custom segment using the mouse, which is possible after using the button “Set graphically ...”.

Edit Operation Parameters.

The Operation Parameters dialog can be reached most easily by double clicking it's line in the Tree. Alternatives are the Parameters menu and the context menu (right clicking in the Tree). The **Operation Parameters are the actual milling parameters**, so the settings for the milling process. Most important settings are the Cutter and the Precision, which can both be found on the first tab page: General..



Which **Cutte**r is best depends on the geometry of the model. The perfume bottle has a freeform outer surface, for which a ball nose cutter (tip of the cutter is half a sphere) gives the best surface quality. It also contains some small details, so a thin cutter is needed. We suggest to use a ball nose cutter of 4 mm diameter (Radius of the ball nose is 2 mm). The suggested cutter for inch users is the 1/8" ball nose cutter, so with a radius of 1/16". You can choose a cutter by its name using the combo-box button at the right of the current cutter name. To look at the dimensions of each of all available cutting tools use the option Library of Cutters, in the Options menu. This library is where you can define a new cutter, or modify an existing one to match your real tool.

Equally important are the **Precision** parameters. They determine the accuracy of the model, and also the time needed for both calculating and machining. For a first rough prototype of the bottle the default value of D/9 will be OK, where D is the Diameter of the cutter. For a nice smooth final model a smaller value is needed.
The meaning of Distance between toolpaths (also called stepover) and Stepsize along toolpath has been explained in the previous lesson. It is recommended to enter equal values for both settings, though for special cases you can experiment with different values for these parameters.

DeskProto offers predefined values for the Precision parameters. These are dependent on the cutter dimensions, and make sure that the DeskProto algorithm achieves the maximum possible accuracy. You are free to enter other values as well, however, DeskProto will always round the value to the nearest "Diameter of cutter / odd number".

A rough estimation of the machining time for your current settings is given by the command "**Estimate Machining Time**" in the Create menu. Do note that the estimation is indeed rough: see the reference manual or the Help file for more information on why it is rough and on how it can be calibrated.

The default spindle speed (rotation speed of the tool in rpm) and Feed (traveling speed of the cutter) will in most cases be correct as well. In fact optimum values depend on the type of material you want to cut, however, when cutting light materials this is not critical.

In this lesson we will just skip all other Operation Tabs, as suitable default values are present, and continue. More is explained about the other parameters in the next lessons, in the Reference manual and in the Help file.

Calculate Toolpaths

Just as in lesson one, you can now start the milling calculations by pressing the button Calculate Toolpaths. Alternatives are: Create menu >> Calculate Toolpaths, Create >> Write NC Program (which detects that the toolpath has to be calculated first), and the Subjects in View dialog box where you can make the Toolpath active in order to start the calculations. Calculation will be quick, and the resulting toolpaths will be drawn in red lines.

You will see (in case you did use cutter Ballnose R2) that two different horizontal layers of cutter movements have been calculated: the first at level Z = -15 mm, the second at the final depth. These are Roughing layers: the cutting length of this cutter is 15 mm, while the bottle half is higher. DeskProto detects that the cutter cannot cut this depth in one go, and inserts an intermediate layer to be machined first.
You may also see many horizontal dashed lines in gray, drawn over the geometry. These are positioning movements in Rapid mode. In the second layer the cutter will skip all area already done in layer one, using a positioning movement above the top of the block. These positioning moves are not present when the optimization option **Sort** (Movement tab) has been applied. In the Entry edition Sorting is always done.

Note: attentive readers will have noticed that the height of half a bottle is in fact only 14.44 mm, so less than 15. They also will have seen that the toolpaths



go below the minimum Z-value of the segment. Draw a side view and compare the green line of the segment to the red lines of the toolpath. The explanation is that when using a ballnose tool DeskProto will always go the Radius of the tool below the minimum model dimension (so in this case 2 mm deeper). This is needed in case of (almost) vertical walls, which otherwise could not be machined completely. This is of course very important to remember when starting the milling machine.

In fact the use of layers as just mentioned is a type of Roughing functionality, which you had not selected. However this basic Roughing is always present, in order not to damage machine or tool: for the first operation DeskProto does not allow the cutter to go deeper than it's cutting length.

You can now save the NC program file and continue with paragraph "To the milling machine", or you can add an extra Roughing operation first.

Optionally add a Roughing Operation

Roughing is quickly removing most of the excess material, using 'rough' settings (a large Distance between the toolpaths). When several layers are needed this is much quicker than doing all layers with the fine toolpath distance needed for finishing. A second advantage is that when finishing after a roughing operation the cutter does not need to remove much material, so it will not vibrate and the result will be a very smooth surface for the model.

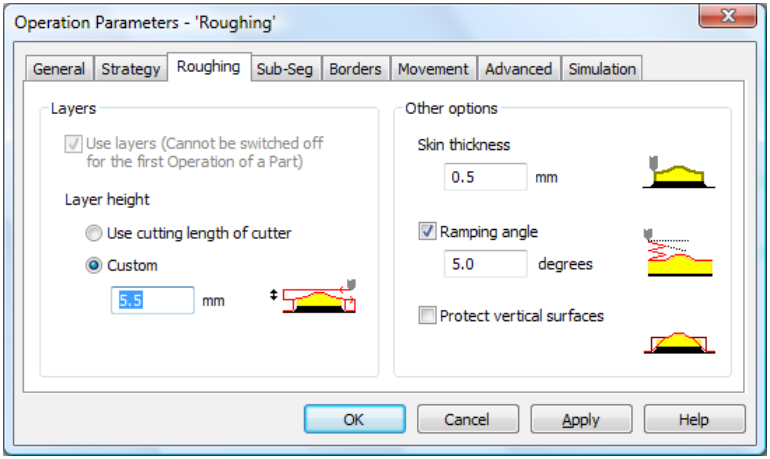
In order to add roughing to the toolpath in DeskProto you will need to add an extra Operation to your part. The first operation then can be set for roughing and the second for finishing. It is of course most efficient to use a thick cutter for roughing as that can remove material quicker than a thin cutter. However, in this case the material that needs to be removed is not much so you can also use the same cutter for both operations. The advantage obviously is that you then need not change cutters halfway the project . Unless of course you have a machine with Automatic Tool Changer (ATC), then this advantage does not count.

So first you need to Add an Operation. This can be done by right clicking the line "part" in the Project Tree and then selecting Add Operation in the context menu. Or as an alternative you can Add or Copy Operations in the Part parameters dialog.

The new Operation's line is automatically set in edit mode, so you can change it's name from "operation [#1]" to "Roughing". If this did not work then right click on the operation's line and choose Rename. The same way you can Rename "operation" to "finishing". These names are not used in the NC file, still using proper names is recommended to remember your intentions.

Now the sequence of the operations is wrong (roughing of course needs to be done before finishing). You can fix this in the Part parameters: using the black arrow buttons called “Move” on Tab page General you can change the sequence of the operations. In order to do this you have to select an operation (make its line blue) first. Alternative is to use the context menu in the Project Tree, as shown in lesson 1.

Note that after this change the toolpaths for Operation Finishing will still show the layers (switched on automatically when it was the first operation). Please uncheck the option “Use layers” for this operation. The toolpaths then will be recalculated, now without layers.



Next you can set the parameters for the Roughing operation. Open the Operation Parameters dialog for this operation (double-click the correct line n the Tree). On tab General you need to select the correct cutter: as said before you can use the same 4 mm diameter ballnose cutter as used for finishing. Then you can add the actual roughing parameters, on tab page Roughing:
- You can set the **Skin** (to be removed during finishing) at 0.5 mm (0.02”).
- The **Layer height** thickness can be fine-tuned: select Custom and enter a value. For light materials like foam or tooling board you can enter 5.5 mm (0.22”) to have three layers that are about equal. For stronger materials like perspex or metal you will need to use a smaller value.
- **Ramping** concerns how the cutter travels to the first point that needs to be machined. Standard the cutter first travels to the correct XY and then just plunges into the material, moving along Z. When ramping this drilling move is replaced by a declining movement (see the illustration in the dialog), for which you can set the gradient angle. This will make cutting much easier than in a pure drilling movement, especially when machining metal.



Do not forget to also choose new Precision values (tab General): both the Toolpath distance and the Stepsize can be set to D/3 (1.67 mm or 0.055") , in order to quickly remove the material.
Here again attentive reader may need some in-depth information: the dialog mentions D/3 for the 1.67 mm value, while the cutter has a diameter of 4 mm. What happens is that after setting a Skin DeskProto will calculate using a *Virtual cutter,* that is the Skin thickness larger in all directions. You are welcome to again forget this detail as this will be done fully automatically.

Now you can again use the command Calculate Toolpaths to also have the Roughing toolpaths calculated. The resulting view will be rather a mess of red and gray lines. It is easy to make it less confusing: in the Project Tree you see a **yellow light bulb** on each line. Clicking the light bulb for an Operation will make it gray (the light is switched off): this will make that operation invisible. So it is easy to view only the roughing toolpaths (make finishing invisible) or only the finishing paths.

Three more detail settings can finally be used to fine-tune the results.

1. For Roughing it is more efficient to choose a different **Strategy** (second tab page of the operation parameters): when you select strategy Block instead of Parallel you will see that less positioning moves are needed.

2. When Finishing it is not needed to also machine the flat area around the bottle model: the material there has already been removed, and this **Ambient** area does not need to be finished. On the Advanced tab page of the Operation parameters you can select "Skip hor. Ambient". You will be able to see the difference immediately after pressing OK.

3. On the Movement tab of the Operation parameters you can reduce the **Feedrate for high chiploads**. This is a great option: it will make the cutter move slower when it has to machine at it's full width. Normally the cutter removes only a thin slice of material when cutting, however for the first toolpath or when entering a hole in the part it may have to remove much more material: a high chipload. DeskProto can automatically detect these situations and then reduce the feedrate to the percentage given here. For roughing in tooling board you can set this to 30 to 50%, for materials like perspex or aluminum even lower.

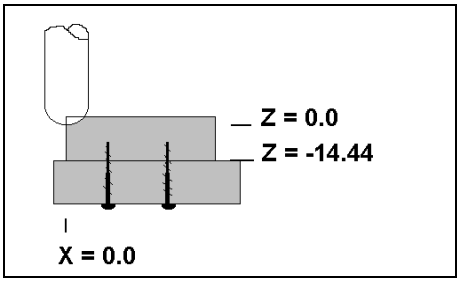
Finally you can save the NC program file. Note that when both operations use the same cutter (and are both visible) DeskProto writes one combined NC program file. When you have selected different cutters DeskProto will write two separate files (unless your machine has an Automatic Tool Changer). You can also force separate NC files by making operations invisible.

To the milling machine

After writing the NC program file you can send it to the machine to create half a bottle. In Lesson 1 you have learned how to fix the block of material and where to locate the WorkPiece zero point (the 0,0,0 position). For this bottle model the process is almost the same, though a new trick will be introduced to get two halves that exactly make a complete bottle.

The idea is to machine exactly half a bottle, the flat bottom surface of the block being the plane of symmetry to be used to joint both halves. Fixturing faces the same problems as described in Lesson 1: the cutter will cut on all sides of the model, and the tip of the (ballnose) tool will come **below the bottom of the block** and may damage your machine's working table. The solution is the same as in Lesson 1: use a slab of wasteboard below the block, and connect both using a few screws from below.
Note: in case of a light type of material like PUR-foam screws are not needed: use double-sided adhesive tape to attach both blocks to one another and to fix them on the machine table.

Make sure that:
- The lower block has its top plane and bottom plane exactly parallel.
- The top block has a really flat bottom surface (needed for the gluing afterwards).
- The screw tips in the upper block are well within the portion of the prototype that will remain after all milling is done. Otherwise either the prototype will fall off during the milling (in case the screws are completely outside), or the outer surface of the prototype will be damaged (in case the screws are too long and their tips will be machined off).



Now you can fix the blocks on the machine, and enter the (0,0,0) position. For X and Y this is done just as in Lesson 1, for the Z this is different. While in lesson one you entered Z=0 on the top of the block, now you start with the bottom of the block (which is the symmetry plane of the bottle). Position the tip of the tool on the same height as the bottom of the block. From there go up 14.44 mm (for inch users: 0.57") and set the Z=0 level there. You can read this



value 14.44 from in the DeskProto Geometry Information dialog box: Tab 'Part' shows that the minimum Z of the prototype is -14.44 (the symmetry plane) and that the maximum Z is 0.

The real Z-level of the top of the block is not very important now: it is OK as long as it is not below Z=0. Obviously it must not be too high either, as the cutter must be able to remove the extra material above the model top. If needed you can solve the latter problem by setting the maximum Z Segment dimension of the part to a higher value.

Now start cutting and create half a bottle. Repeat the complete milling operation using the same NC program to get a second half. Glue the halves together and your model is ready !

Lesson Three

The Venus Bust (rotation axis)



In the third lesson you will learn how to create toolpaths for a rotation axis: an extra item on your CNC milling machine that lets the object rotate during machining. It is also called A-axis, and looks like a spit on your barbecue. This lesson of course only is useful in case you have a machine with such an axis. Rotation axis machining is available only in the DeskProto Multi-Axis edition.

A model will be created of the famous Venus of Milo statue (The Louvre, Paris), or in fact of its head only. The geometry has been scanned on a Minolta 3D scanner, and was exported as polygon data. We may use this geometry data by courtesy of Minolta Corporation in the USA. Because of its size (7 Mb) the file venus.stl is not included in the standard DeskProto setup. You will first have to get the file: copy it from the Bonus geometry section of the DeskProto CD or download it from the DeskProto website.

Start a New Project

This third lesson will again start with creating a new project, so that you can again take all steps needed to go from geometry file to NC file. Only the steps that differ from the previous lessons will be explained. In addition to following this lesson you can use the Rotation axis milling Wizard to show you all steps to be taken. You can also view the tutorial videos on the DeskProto website.

The geometry file that you have to load is called **Venus.stl**. Note that this file is not installed in the Samples folder. You will first have to copy the file from the DeskProto installation CD (section Bonus geometry), or download it from www.deskproto.com The file is only available in mm, so inch users will have to scale down with factor 0.03937 (Part parameters, tab Transform), or in fact a smaller factor in order to create a scale model.

The rotation axis on most machines is parallel to the X direction, and is then officially called the A-axis. On the one side the rotary table's vise or chuck holds the workpiece, and often on the other side a tail-stock is available to support the far end. For instructions on how to use a rotation axis parallel to Y (a B-axis) search the forum on the DeskProto website.

When you view the Venus geometry after loading, first thing that you will observe is that the geometry is not orientated correctly. The most logical rotation axis for a head is the line from neck to top (so a vertical line when standing upright). This line now is along the Y-axis instead of along X as needed, so you will have to **rotate the geometry minus 90 degrees around Z**. When you check the model's dimensions using the Geometry Information dialog, you will also see that you have to **scale down** (so use a scale factor between 0 and 1) as the model is full size and so will not fit most milling machines..

With continuous Rotation axis machining (as explained in this lesson) the part rotates during machining: XZA toolpaths, only the X, A and Z axes are used. The Y-axis does not move: the cutter needs to be positioned exactly above the rotation axis when starting to machine.
The alternative use of a rotation axis is **Indexed machining**. Here plain XYZ toolpaths are applied, from different sides, with a positioning A-rotation in between. DeskProto supports indexed machining in the wizard called *One or more sides, automatic rotation* (The "N-sided wizard"). That wizard will automatically generate a number of parts: one part for each side to be machined.



Use the Rotation axis

The first thing to do after loading the geometry is to switch on the option **Use Rotation axis**. This option can be found on the first Tab of the Part Parameters dialog box. Of course this option is available ONLY in case you selected a machine with such optional 4th axis. This means both that your actual machine must have one, and that in the DeskProto machine definition in it must have been configured. So if this option is grayed out then go to the machine library (Options menu) to fix this in the machine definition. In case this option is not present at all then you are probably not using the Multi-Axis edition of DeskProto.

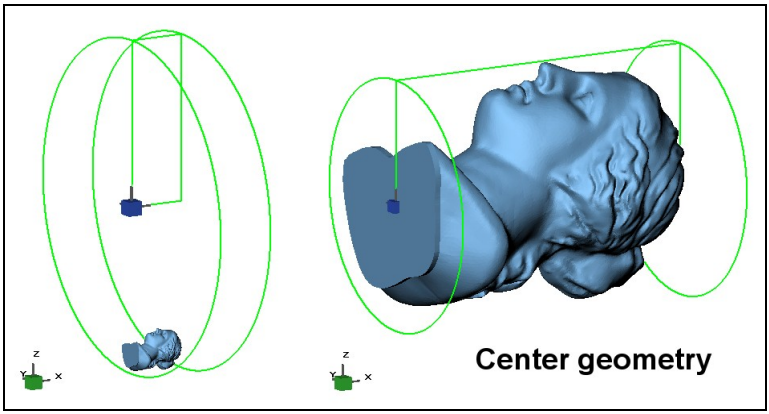
When you check the option Use Rotation axis a message will pop up. It tells you that a few parameters will be automatically changed to accommodate rotation axis machining. Most of these parameters will be discussed below.

After switching on the Rotation-Axis and pressing OK (and probably acknowledging the size error, see below) you will see that a completely different segment will be drawn on screen. Remember: the part segment is the block of material to be used. No longer the rectangular block (in green lines) that you are used to, but instead a cylindrical segment. This does of course make sense for rotation axis machining. DeskProto does not support a block segment for rotary machining.

Two different size errors are possible here. The first is that the model does not fit inside the working area of the machine. For Venus this does make sense, as the unscaled geometry is 284 x 421 x 315 mm. So you will have to **scale down**, the scale factor depending on your machine's dimensions.

The second size error is that the part cannot be rotated as it would collide with the machine's working table. On your machine the rotation axis will be located above the working table, and the radius of the cylinder segment may not be larger than the distance between table and rotation axis. Otherwise of course the block cannot be mounted. This distance can be set in the machine definition (Library of machines). DeskProto performs this check for cylindrical stock: if you use a rectangular block you might have to cut off the corners to make rotation possible.

This second size error also may pop up when the actual part is not too large, caused by an incorrect calculation of the part segment (the cylinder block of material). See the illustration below.



The object is rotated round the "real" X-axis (so the axis at position Y=0, Z=0). For the Venus STL file this rotation axis is located outside the geometry, resulting in a very large cylinder being drawn as the part segment, and causing this second size error. An extra option is present to solve this: on the Transform Tab of the Part Parameters, check the option **Center geometry**.
Now (at last) DeskProto will accept Apply and OK without errors, and the segment drawn on screen will be the one that you need.

An important issue is the position of the **WorkPiece Zero point**. When rotation axis machining you can change the X-position of the zero point as usual, using the Translation option, however the Y cannot be changed (Y is not used) and for the Z fewer options are available. Two different conventions do exist for Z: the workpiece zero point can be either on the rotation axis (so inside the block) or on the outside surface of the (cylinder) block. So for Z you can choose between "Make top of part zero", setting the Z=0 at the highest point of the cylinder, and "None" which will set the Z=0 exactly on the rotation axis.

As said in the message-box just mentioned DeskProto has already set the Z Translation to None. The result can be seen in the Geometry Information dialog (Tab page Part) and (easier) in the position of the **blue cube** (the "Orientator on the WorkPiece zero point") on your screen. Make sure to set this WP zero point on your machine accordingly: an incorrect setting may damage part, cutter and/or machine !!

Two other parameters that have been changed automatically:
The **Part segment** has been set to Upper half only: as the geometry is rotated the bottom half will become top half after 180 degrees, so upper half is sufficient to machine the complete model (for most geometries).
The **Borders** have been set to 'No extra', for all operations. You can find this



parameter on the Borders tab of the Operation parameters. Its normal value is 'Extra for cutter', as normally all outside surfaces of the model need to be machined. For rotation axis machining the model needs to remain connected to the rotation axis unit and to the tail-stock. So the outside surfaces left and right may not be machined, which is achieved by this setting.
For some geometries it will be handy to add **Support blocks**: extra material left and right, for a better connection to rotation axis unit on the left and to the tail-stock on the right. As the Venus geometry has a flat bottom (now on the left side) such blocks are not needed for this project.

When the rotation axis is used, for some parameters in DeskProto the Y-axis setting (to be set in mm) will be replaced by an A-axis setting (to be set in degrees). Do note that DeskProto replaces Y by A, so in fact what you do remains 3-axis machining. You can choose to use either XYZ (normal) or XAZ (rotary), DeskProto will not use all 4 axes simultaneously. Still DeskProto offers you full 3D use of the A-axis: not just wrapping some flat 2D toolpath (sized 360 mm) around a cylinder by simply replacing Y by A, but real 3D toolpath calculations around a 3D CAD geometry.

The use of an A-axis does influence the Min and Max segment coordinate values. For XYZ machining these values simply define the outer edges of the segment block. For rotation axis machining this is different.

- The **X** values define the length of the cylinder (which is the same as for XYZ machining).
- The **A** values have to be set in degrees between 0 and 360, permitting you to machine a wedge-shaped section (a piece of pie) instead of the full cylinder.
- The **Z**-values by default will have been set to 'Upper half only", as just explained. A Min Z above 0.0 will result in a ring segment, a Min Z below 0.0 will result in the tool machining below the rotation axis.

In review, in order to prepare the Venus geometry for machining the following settings have been made:
- Load geometry Venus.stl
- Check "Use rotation axis"
- Rotate -90 degrees around the Z-axis
- Check "Center around rotation axis"
- Scale down the geometry until it fits in your machine.

Fine-tune the parameters

Basically the DeskProto parameters can be used for rotation axis machining exactly as they are for standard XYZ machining. There are a few exceptions to this, which will be explained in this paragraph.

As said before, all parameters concerning the A-axis are in **degrees** instead of in mm. This does not apply to the precision settings though: for toolpaths in the X-direction the Distance between toolpaths should be in degrees, for toolpaths in the Y (A)-direction the Stepsize should be in degrees.
However, as degree values are difficult to imagine, DeskProto hides these, and asks for values in mm or inch instead. These will be converted to degrees at maximum segment radius, so at the outside of the cylinder, after which DeskProto will internally use the degree values. This means that the actual mm values that are used will in most cases be smaller than the values that you entered, as most movements are closer to the rotation axis.

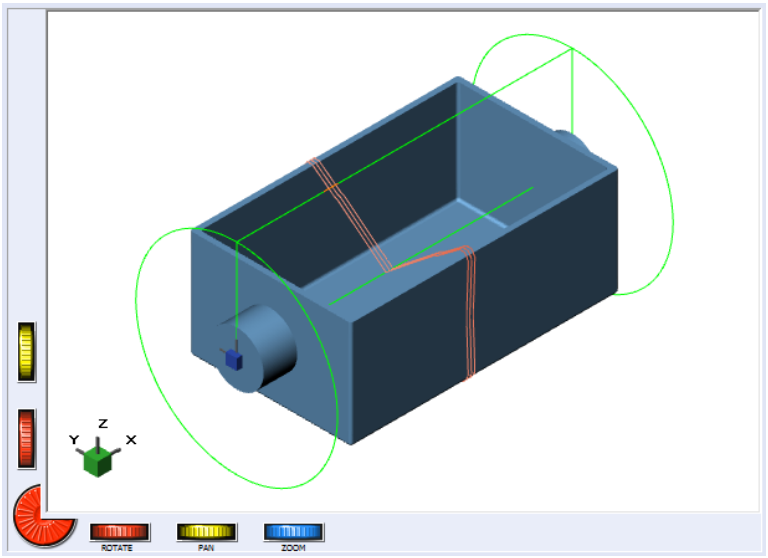
With XYZ machining the tool cannot reach the bottom side of the model, so during machining the part will remain resting with its bottom on the working table. For rotation axis machining the part **fixturing** is more complicated: it is possible to completely cut your model loose from the machine, making it fall down during machining. The resulting model then will both be damaged and unfinished. DeskProto will take care that this does not happen, by assuring that some connection will be present between the rotary table and the model, or rather by popping up an error dialog if such connection is not present.

A valid connection can be accomplished in several ways. For models with a flat side, like the Venus head in this case, you can make sure that this complete side surface remains un-machined (and thus connected to the rotation axis vise). This can be done, as explained above, by switching off the borders (Operation parameters, tab Borders).
Next you can decide what to do with the right side (top of the Venus head): machine it or not, depending on whether you want to use a tail-stock or not. You can do so by changing the max X value of your segment. In fact you then manually add a border area at that one side only.

Without such flat surface you may want to define a **connection shaft** to connect the part to the vise: a cylinder shape where the cutter may not come. Easiest way to do this is by defining one or two cylindrical support blocks (the default supports will be cylinders when rotation axis machining has been selected). Note that then the Borders need to be switched off too, in order to prevent machining the flat outside surfaces of the support blocks.
An alternative way to define such connection shaft is by using the minimum Z Segment dimension. In case this minimum Z is larger than 0.0 a torus shaped segment results, and thus also a shaft (the inside of this torus). When using this method the Borders may be set to “Extra for cutter” as the shaft won’t be

machined anyway. A disadvantage of this method is that also within the model the cutter cannot go below this minimum Z-level, so for certain models some area will remain not machined.

For this Venus lesson just make the Max X segment dimension a bit smaller to prevent the cutter damaging your tail-stock - of course only in case you use one.



When rotation axis machining the **Z-segment boundaries** do require extra attention: both Z-min and Z-max. As just said: the **Z-min** has been set to 0.0 (Segment choice "Upper half only"), as for most geometries the cutter does not need to travel lower than the rotation axis.
This is not true for all geometries though: for instance when machining an empty box, see the illustration above. It will be clear that for such geometry the cutter needs to travel below the rotation axis in order to machine the inside of the box, so to a Z-level below 0.0 Thus, you have to choose a Min Z value below Z=0. In such cases do take special care then when setting the workpiece zero point on your machine: this has to be done VERY accurately, or you will see marks where the toolpaths cross the Z=0 level.

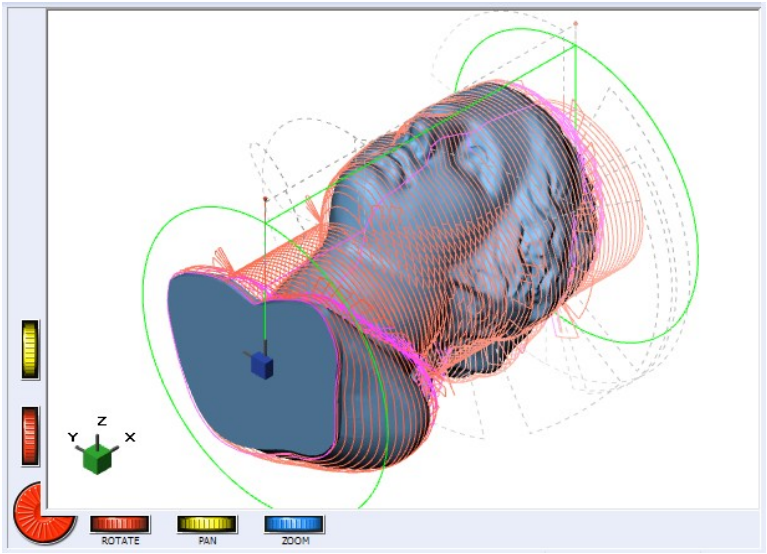
The screenshot also illustrates the presence of **undercuts** for rotation axis machining. Undercuts are areas where the cutter cannot reach. For XYZ machining this is the bottom side of an object. For rotation axis machining this is different: see the three red toolpath lines drawn in the picture above. Only the X, Z and A axis will move: the Y-position is constant, with the cutter

positioned exactly above the rotation axis. This means that the cutter cannot completely empty the box: it just cannot reach all four vertical walls on the inside of the box. The V-shaped toolpaths that are drawn illustrate this. For this geometry it will be better to machine from two or from four sides, also using the rotation axis: indexed machining.

The **Max-Z** segment boundary requires extra attention as well. In many cases you will prepare your block of material using a band saw machine: resulting in a rectangular block. The part to be machined has to fit within this block, and so the block's maximum diameter (the diagonal line of the block) may be much larger than the diameter of the cylinder calculated by DeskProto. This may result in overloading and damaging the cutter. You can easily prevent this by setting the Max Z segment boundary to a higher value. DeskProto will then add extra layers as needed to remove the extra material at he corners of the block. Some of the toolpaths then will be cutting air, still better than breaking your cutter.

Finally: perhaps you want to use toolpaths in the A-direction, combined with either conventional or climb machining. This way the rotation axis can remain rotating in the same direction (**continuous rotation**), without the need to travel back to the other side of the model for each next toolpath as with XYZ machining. Nice idea, however: most rotation axis units can only keep rotating in one direction for a limited number of times. As DeskProto does not know how many (on some machines this is just 3), it will not allow this continuous rotation. Instead after each toolpath the tool will move up to Zfree safe height and the rotary axis will rewind the full 360 degrees.
When your machine does support continuous rotation in one direction you can overrule this setting. In the machine definition (Options menu) check the option "A-values may exceed 360". Then the axis will keep rotating in the same direction - unless of course the milling direction is set to Meandering (Operation Parameters, tab Movement), in which case the machine will keep cutting in both rotation directions.

So far the background information for rotation axis machining, most of which is in fact not needed to complete this Venus model but may prove to be useful later.



The resulting toolpaths will look like in the picture above, depending on your choices for scaling, cutter, precision, etc.
The picture shows a Roughing operation. You can clearly recognize a series of circle toolpaths forming a cylinder: the first layer. The gray dashed lines are the positioning moves in the lower layers, skipping any area that has already been machined (if Sorting is off you will see many more of these than in the illustration above). The bottom surface of the statue remains un-machined: the connection with the rotation axis. The toolpaths shown are in A direction and meandering, as you can see on the forehead.

Just as for plain XYZ machining it is advised to do a Roughing operation first and then a finishing operation. The rotation axis wizard will do so automatically for you. For more information you can watch the Venus tutorial videos on the DeskProto website.

To the milling machine

Again it will be clear that the instructions given here cannot be very accurate, as a detailed set of instructions would be different per machine. Still a number of appropriate general remarks can be given.

First you have to set the **WorkPiece zero point**. With the default translation settings this will be on the left side of the cylinder block (X=0), with the tip of the cutter exactly on the rotation axis (Y=0, Z=0). When you model is not round then also the A=0 maybe important.
Setting X=0 and A=0 is easy: move to the correct position and tell the machine to use that as zero. Setting Y and Z is more difficult as it is not easy to see if the tip is on the correct location. Some machines offer a calibration help, like a horizontal surface at the correct Z-value. If not you need to visually find the zero positions as accurately as you can.

A good help is to machine a small calibration cylinder: fixture some material, mount a flat tipped cutter, and switch the spindle motor on. Then move the cutter to Y=0 and Z=+10 (uncalibrated) and next perform a full rotation (A 0 to 360 deg). The result will be that you have machined a cylinder. Now measure the diameter of the cylinder: it should be 20 mm. If it is not then you can correct your Z=0 position accordingly. Say it is 21.4 mm: 1.4 too much, meaning that the current Z=0 is 0.7 mm too high.
You can then use the same cylinder to calibrate Y=0: let the cutter touch the cylinder both on the front side and on the back side and write down both 'touching' Y-values. They should be identical but for the minus sign. If not, then correct your Y=0 accordingly.

This is a lot of work: the bright side is that you need to do this only once: the same zero point will be valid for a next job as well.
Note that with this zero position you must take care not to let the cutter travel to the workpiece zero point (which is a standard command on many machines), as this will damage your cutter and/or your material.

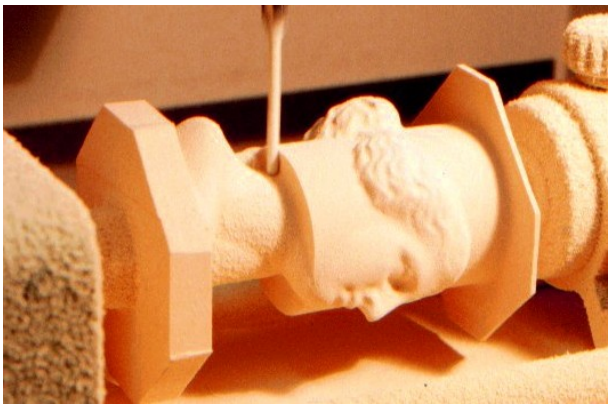
When you have used translation method “Make top of part zero”, the Z=0 needs to be with the tip of the cutter touching the top of the cylinder material block. Now it is critical that the cylinder block has exactly the correct diameter: otherwise the resulting part will be too thick or too thin.

Next you will have to **fixture** your material block, using the available options of your rotation axis, like a 3-jaw chuck, pins on a circular plate or a drill-head. Use the tail-stock with a centering pin on the opposite side if one is available, as this makes the stability of the model much better. For machining rings (in wax, for investment casting) you have to use some special fixturing tool, or use a hollow wax bar that can be fixtured in a 3-jaw chuck.



A detail that you need to check only once is the **Rotation direction** of your A-axis: will it rotate clockwise or counterclockwise for a positive value of A. We have found that no clear standard direction exists: each manufacturer makes his own choice. So the DeskProto default will be correct for about half of the machines. If not (the resulting part then will be a mirrored image of the CAD file) you can correct this in the DeskProto postprocessor: Options > Library of postprocessors > OK on warning > select your post and press Edit > Tab Movement > add a minus sign to the value in edit box “Factor” in column A.

Many controllers have problems in setting the correct Feedrate when rotation axis machining, as the rotation speed needed to achieve a certain linear speed depends on the distance between the cutter and the rotation axis. So take care in setting the Feedrate, and be advised that the machining time estimation may be (very) incorrect. For more information see the FAQ on the DeskProto website.



Finally: before starting the NC program file from DeskProto, do make sure that your cutter is positioned on Y=0. As said in fact DeskProto remains 3-axis CAM software, so the XZA toolpath file from DeskProto does not contain any Y movement command. Y has to be correct before starting.

The machining process will look like the photo above. Do note the cylinder form created by the first layer. After machining you can remove both sides of the block using a simple band saw

For a picture of the result see the next page.



Lesson Four

The Cell phone (two-sided machining)



The fourth lesson is about machining a model from two sides. In DeskProto this is very easy, as the **Two-Sided Milling Wizard** will guide you through this complex process. Because of this wizard this lesson does not need to show all details: it will only illustrate and explain what the wizard will ask you. The two-sided milling wizard is not available in the DeskProto Entry edition.

The example used is (again) a very fine geometry: the front panel of a cell phone. It is not from an existing phone, but has been specially modeled for this lesson by John Brock of Robert McNeel & Associates, using the Rhino 3D CAD software that McNeel manufactures. Also the rendered picture above is courtesy of Robert McNeel & Associates.

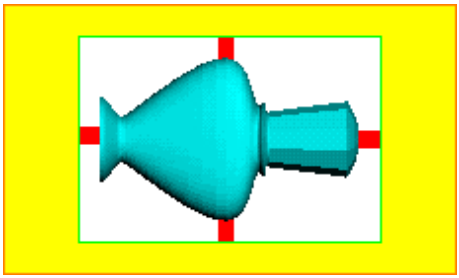
The cell phone STL is a large file (8 Mb), so it is not included in the standard DeskProto Samples. You will have to get the file from the Bonus geometry section of the DeskProto CD, or download it from www.deskproto.com

Two-sided machining

In this fourth lesson you will use DeskProto's Two-Sided Milling Wizard to correctly set all parameters needed to machine a part from two sides. The wizard is meant to be very easy to use, even self-explaining, so in theory this lesson is completely superfluous. Still in practice the process is quite complex: so we felt that illustrating and explaining what you will need to do will make things much easier for novice users.

Start the DeskProto wizard, to be found via the Start screen or via the File menu. Take care to start the 'normal' wizard: the "custom wizard" is meant for some special applications. On the first screen (called "Which wizard") select the wizard named **Two sides, manual flip**.
The alternative for two-sided machining is using a rotation axis for an automatic flip. The advantages of the manual flip are that no rotation axis is needed, and that the block is more stable as it lies on the machining table.

This wizard is a sequence of 7 pages, as indicated by the tabs on top. In addition to this first page (which wizard), these are:
2 What to machine
3 Material and Supports
4 Roughing
5 Finishing
6 Contouring
7 Send to machine.



Generally speaking, when milling a model from two sides the problem is the second side: how to fixture the model in the correct orientation for that side, and how to re-establish the WorkPiece Zero point. The DeskProto Wizard solves this by using an oversized material block, so large that a complete frame remains present around the part. Four small blocks are added as "support bridges": these connect the part to this frame and can be manually removed later. See the illustration above. Because of this frame it is easy to re-fixture the model for the second side, and the wizard also uses the frame to set a repeatable workpiece zero point.



The two-sided Wizard

On page **What to machine** (*Page 2*) first load the geometry file Cellphone.stl. Note that this file is not installed during setup. You will first have to copy the file from the DeskProto installation CD (section Bonus geometry), or download the file at www.deskproto.com. The file is only available in mm, so inch users will have to scale down by a factor of 0.04 (more exactly 0.03937). Load the file using the Browse button on this wizard page.

Two more options are offered on this wizard page: Scaling and Rotating. For metric users **Scaling** is not needed as well (unless of course your machine is too small). As said inch users have to apply a scaling factor of 0.04 (uniform scaling). For the Cell phone the **Orientation** may remain as default, with the top surface on top: rotating the geometry is not needed. Note the light yellow question mark icon, left of each question: it provides help on the question, and also tells you where that same setting can be found in the dialog based interface (so without the wizard).

Page **Material and Supports** (*Page 3*) will add the support blocks (as explained above) and will set all segments. The wizard will select default bridges in order to add four **Support blocks**, at the minimum and maximum X and Y values of the part. The thickness of these bridges depends on the size of your model, their length depends on your default cutter (the bridge must be long enough for the cutter to move around the model). When you later select a thicker cutter the wizard will warn you about that. For more control you can also select Custom bridges, and use the Detail settings to define your own support blocks.

The **Dimensions of the material block** are set by the wizard: both right and left 25 mm (1") is added to geometry + supports, on the front and back sides 10 mm (½") (You can change these values in the Preferences). On the right and left side the frame is made wider as there room is needed for the clamps to fixture the block on the machine. The wizard lets you set only the Z-dimension here: the block's thickness. The value that you enter has to exactly match the actual block thickness, otherwise the two sides won't match correctly. In practice this is easy, as most modeling boards will be delivered in slabs of an exact thickness. The X and Y dimensions of the physical block may have some oversize.

The **Milling depth** is in fact the minimum Z for the Operation sub-segments. The same depth will be used for all operations. Default depth is halfway down the block: for the cell phone geometry a higher depth value is needed in order to completely machine the geometry. Also: do not forget to change the depth after changing the block thickness, as the depth is relative to the top of the block.
Dimensions and depth will be drawn on screen as line drawings: orange lines

for the block, green lines for the area to be machined. These are in fact the DeskProto segments, that you have seen already in the previous lessons. For the total block the part segment is used, for the area to be machined the operation sub-segment.

Page **Roughing** (*Page 4*) offers the setting for the Roughing operation. The first is the **Use** checkbox: roughing is optional, and you can skip this operation by un-checking this box.

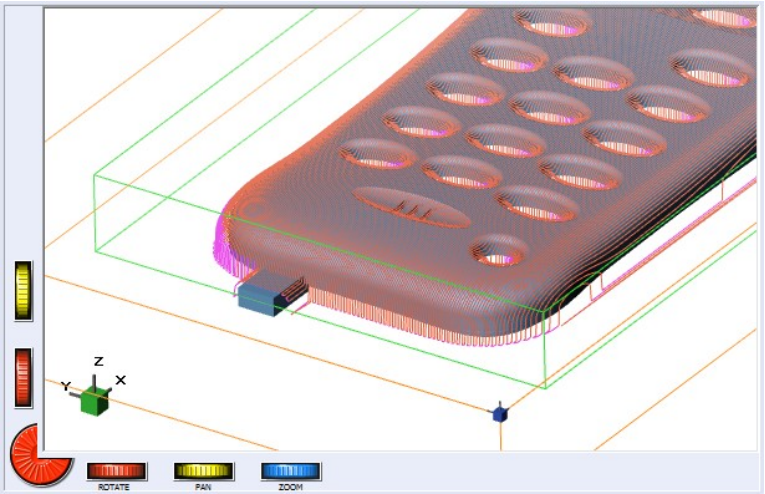
Which **Cutter** to choose is an optimization between many factors:
- cutting speed (a larger cutter can remove material more quickly)
- geometry characteristics (for freeform surfaces use a ballnose cutter, for horizontal and vertical surfaces use a flat tipped cutter).
- surface quality (a larger ballnose cutter will create a smoother surface)
- small details (for small inner radii a small cutter is needed)
- height of the model (small cutters are short)
- use the same cutter for all operations or do a tool change halfway the milling process. For this cell phone part we have used a 3 mm ballnose cutter (radius 1.5 mm), for all operations, in order to avoid a toolchange. 3 mm diameter is sufficient for most details here, and the cutter is long enough to machine the complete model. For inch users an 1/8 inch diameter ballnose will be a good choice.

The **Precision** (horizontal Distance between toolpaths) together with the **Layer height** (how deep the cutter may sink into the material) determine how much material is removed per toolpath. The default values of D/5 (precision) and half the cutting length of the cutter (layer height) will be OK for wood and tooling board. The default **Speeds** will do as well. For light materials (foam, light wood) precision D/3 will do, for materials like perspex and aluminum smaller values will be needed. You will have to find your own optimal values.

The default **Block** strategy is optimal for most geometries, as it will minimize the number of positioning moves during the lower layers.
The default **Skin thickness** is 10% of the cutter diameter, which in most case will be OK as well.

The field for **Estimated machining time** is empty: the estimation will be shown after you have calculated the toolpaths using the green **Calculate** button.

Wizard pages **Finishing** (*Page 5*) and **Contouring** (*Page 6*) are very similar to the Roughing page just discussed. Of course without the Roughing parameters Skin and Layer height. Also a different Strategy has been selected. You will use a much lower precision value than for Roughing.



The illustration above shows the Finishing toolpaths for Side 1. In this drawing you can clearly see that the outside face of the support block is not machined: this way the blocks remain connected to the frame. The wizard has accomplished this effect by setting the Borders to 'Stay within segment' (setting "No Extra" would have achieved this as well, but this setting will make the machined cavity the same size for any cutter diameter).
If you see more positioning moves than in the illustration, the difference will have been caused by the option "Always stay low" (Movement tab).

The final page of the Wizard is **Send to machine** (*Page 7*) and it will show you the project tree that has been created: two parts (Side#1 and Side#2) with three operations each. The difference between the two parts is a 180 degree rotation round the X-axis, just as you will later rotate (flip) the block on the machine. In case you want you can rename any of the names after a slow double click on its line in the Tree.

In this wizard *two* buttons are available to Save the NC program file: one for each part. Do choose file-names that indicate which side of the part it is for.
The buttons to directly send the toolpath to the machine are available only if that option has been configured (in the Preferences). Of course only to be used when your machine supports this option.

The **Report file** also is special for this wizard. The report will list all information that is needed on the milling machine to correctly process these two NC program files. This will be explained in detail in the last paragraph. For now just open this file and print / save it for future reference.

Fine-tune the parameters

You have now finished the Wizard and created two NC program files, so you are ready to start your milling machine and create the model. Still it is important for you to know that at this point, after having finished the Wizard, it is still possible to edit any parameter setting that the wizard has made, and even to add any special parameter setting that could not be done in the wizard. Of course after any change you will have to again write the NC program file(s). If you do not need any changes you can also skip this paragraph and continue with the next: “to the milling machine”.

The wizard has created three operations for each side. You may want to add one or more additional operations. For instance a Detailing operation with a very small tool (diameter 0.8 mm or 0.03") for the microphone and speaker slits. Then you can change the operation sub-segment to Freeform, and select two small areas to be machined. For the second side you can also add an operation with a flat tool (2 mm diameter), as the inside geometry contains many sharp inner corners.

When adding operations you have to be careful though: some changes will ruin your two-sided milling setup. What you must NOT change are some setting for the sub-segment (the area to be machined may not be enlarged) and for the borders. The best way to add an operation is by **Copying** an existing operation, as then the settings for sub-segment and Borders will be copied as well. All other operation parameters can be changed as needed.

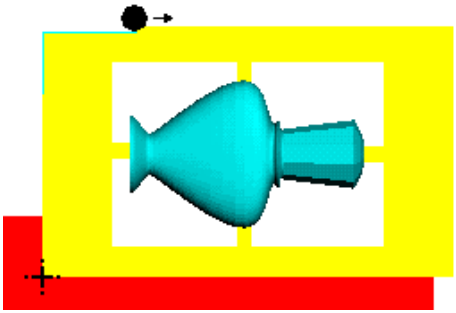
In the Part parameters you must not change the block size (the Part segment), as this size is needed to exactly match both sides and set the WorkPiece zero point. This WP zero point will be the same for all operations in the part. The support blocks may be changed. Obviously the support blocks for both sides must exactly match, and to avoid any differences DeskProto has selected the option “Use bridges of first part” in the second part.

An example of such detailed project file (cellphone.dpj) can be found on the DeskProto CD.

To the milling machine

Normally, it does not really matter where on the machine's working table your block of material is located as you can freely choose the workpiece zero point. It also does not matter whether or not the block is exactly lined up with the machine's axes (as the block will have some oversize). For two-sided milling this is different, as for the second side both position and orientation of the block must exactly match position and orientation of the first side.

For this aim DeskProto uses a **ruler**, mounted on the machine's working table, that is exactly parallel to the machine's X-axis. See the illustration below: the long red 'horizontal' bar is parallel to X, with an **end-stop** at the left side of the ruler. An easy way to create ruler and end-stop is to just machine them using manual control: that way they are sure to be parallel, and their positions are exactly known.



The WorkPiece zero point for the NC program files has to be set at the point where ruler and end-stop meet (the cross-hairs in the illustration above), the Z=0 set with the tip of the tool touching the top of the block. As said this is for both sides: the WP zero point remains the same after turning the block upside down for the second side.

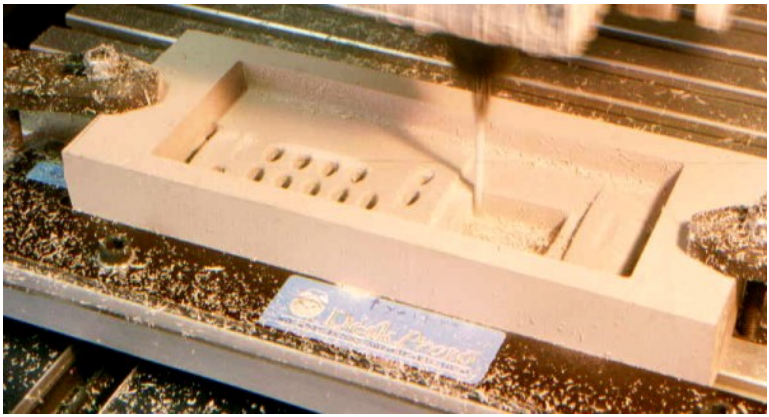
After machining the first side, you have to machine two **reference planes**, on the block's back and left side: see the green line in the illustration above. The reference plane on the left is on position X =0, so at the exact position of the end-stop, and needs not be along the complete side (which would not be possible because of a clamp). The reference plane on the back is on a Y position as specified in the Report file, and needs to be machined along the entire back edge of the part. It needs to be bit deeper than the ruler is thick.

You can machine both reference planes using manual control (mind the radius of the cutter when doing so!). DeskProto does not machine these reference planes automatically, as it does not know where you have placed your clamps.

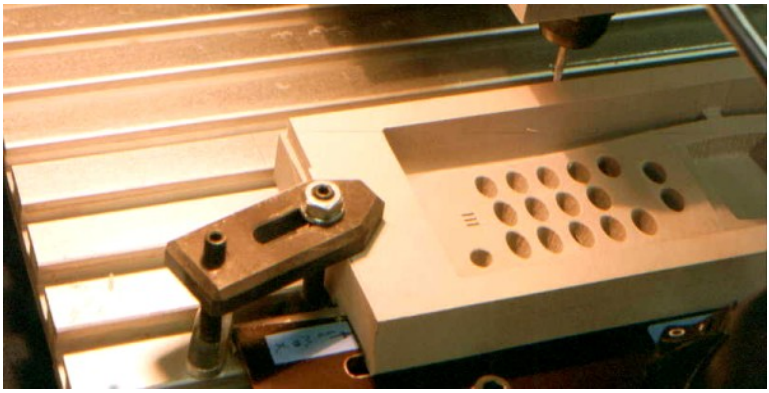
After turning the block upside down (“flipping” the block) you can press these two reference planes against ruler and end-stop, and fasten the clamps. This makes sure that the block is exactly lined up with the machine, and is exactly at the correct position.

In addition to the **Ruler /Reference plane** method just described, several other methods are available to correctly position the block after turning upside-down. For instance using **Reference Pins** on the machining table and drilling holes to exactly fit these positioning pins. Or a rotation axis with a 180 degree rotation around X. The Two-Sided Milling Wizard can be used with any of these positioning methods, as long as they result in the block having the same position before and after turning upside-down

Below you can see a few illustrations that show this process for the Cell phone model.



In this illustration the first side of the cell phone is being machined. Do note the dark brown ruler in front and left, and the two clamps right and left used to fix the block. The cutter currently is machining the second layer.



Before loosening the clamps, first you have to machine two reference planes on the left and on the back, as shown here. This is why some oversize is needed on the back side of the block. Also the left side of the block needs to be a bit inclined to have some excess material to be machined.



The second side now can be machined using the same workpiece zero point: the toolpaths will exactly match.



The resulting cell phone front cover: fresh from the machine, without any extra work done. Note the two machined reference planes and the support blocks (bridges): both clearly visible. The frame and bridges have to be removed manually: some sanding will be needed where the support blocks were attached to the model. The rest of the model will already be smooth enough.

Lesson Five

2D Machining



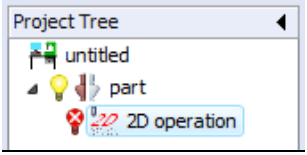
The lessons so far have all been addressing DeskProto's main application: creating 3D prototypes. As additional functionality DeskProto also offers some basic options for 2D machining. You have to be aware that DeskProto is not 2D CAM software: when you need options like 'pocketing' and cutter compensation we advise you to also buy a simple 2D CAM program.

In this lesson you will learn to use these 2D machining options: to engrave text and/or a logo in your prototype, and to use 2D toolpaths to assist in creating the model. In addition to this lesson you can also use the **Basic 2D Machining Wizard** to teach you the basics of 2D machining.

Create a 2D Operation

2D machining in DeskProto is done in a special type of operation: the 2D Operation. In the previous lessons you have only seen 3D Operations, and when you open DeskProto the default project will open which contains only one operation (which is a 3D Operation).
So in order to do any 2D machining in DeskProto you will somehow need to create a 2D Operation.

Easiest way to do this is to use the Start Screen: the dialog that pops up on program start, offering a choice of common tasks (if the Start screen does not pop up you can open it via the File menu). Make sure to check the box "Use samples folder" so that you can easily open a Sample file later-on. In this Start screen you now can select task "Start new project" >> "2D project". You will see that the project tree now has changed: a 2D Operation is present instead of the standard "operation" which is 3D.

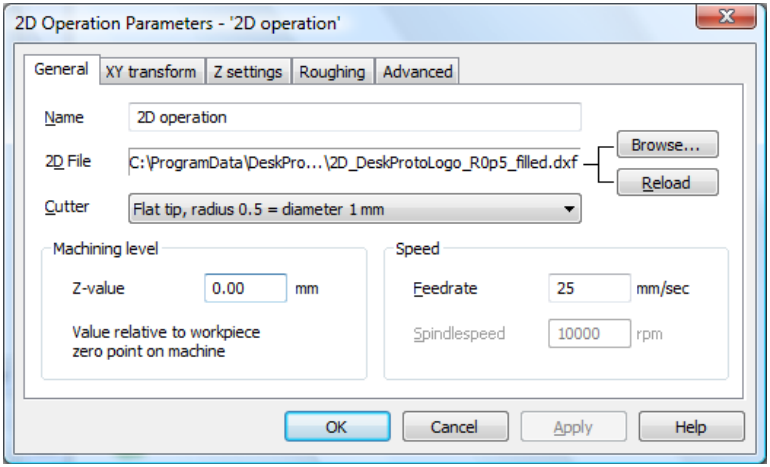


You can also create the 2D Operation manually, by editing the Tree (with the Start Screen closed): right-click on the line "part" of the project tree, and in the context menu that will pop up select "Add 2D Operation". You will see that after adding the operation a new line appears in the tree, with a different icon, showing the new 2D Operation. This line starts in Edit mode, allowing you to give the new Operation a proper name.
As shown in the illustration, the lamp icon in the Tree is in **red** indicating that the operation is invalid. Reason is that you have not yet selected a 2D file for this operation, and without such file a 2D operation is invalid.
After manually adding a 2D operation you can remove the 3D Operation using a right mouse-click on that line of the Tree and selecting Remove).

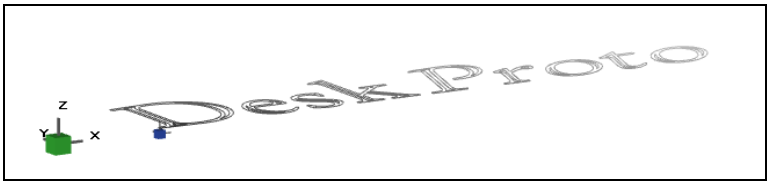
It is also possible to combine 2D and 3D in one project, more about that will follow later in this same lesson.

Open a 2D file

Double-click on the new line in the tree to open the 2D Operation parameters dialog (or use the command Operation parameters in the Parameters menu). The dialog **2D Operation Parameters** will now open, which will look familiar as many of the parameters are identical for 2D and 3D.



In your dialog the entry for **2D File** will be "None". Use the Browse button at the right of the 2D File field to locate and open a suitable 2D DXF file. For this lesson we will use the DeskProto logo file that comes with DeskProto: the file *2D_DeskProtoLogo_R0p5_filled.dxf* or it's inch equivalent. You can find it in the DeskProto Samples folder (see the previous page). The "filled" in the name indicates extra lines to remove all material within the logo. Without changing any other parameter press OK: the result will look like this:



By rotating the view you can check that it is a pure 2D contour line: all lines are on one flat plane at constant Z-level. Note that the red lamp icon in the Tree has now become yellow.

For 2D files DeskProto supports the DXF format and the EPS (or AI) format. Engineering software will typically generate DXF files, while graphics

software in most cases only can do EPS (postscript). Of both file types only a subset is supported:
The **DXF** subset includes point, line, polyline, LW polyline, arc, circle and ellipse.
The **EPS** subset includes point, lineto, curveto and moveto.
DeskProto will convert arcs to polylines. A point in the 2D file will result in drilling a hole on that location. Spline curves will be ignored.

2D files normally speaking do not contain any Z coordinates: DeskProto will draw the 2D contours at level Z=0 and you need to enter Z-values for the toolpaths.
Still, as many users requested that, it is also possible to open a DXF file containing a 3D polyline here. In that case DeskProto will ask you whether or not to use these Z-coordinates. If yes, the Z-values that you enter for the toolpath then will be added to the Z-values as found in the file.

The project that you have created does not contain a 3D geometry (STL-file), as that is not needed for 2D machining. Because of that most Part parameters will have disappeared from this project: the part parameters dialog will only show the General tab page.

2D Operation parameter settings

Now re-open the dialog “2D Operation Parameters” that was just closed, and look at the available parameter settings.

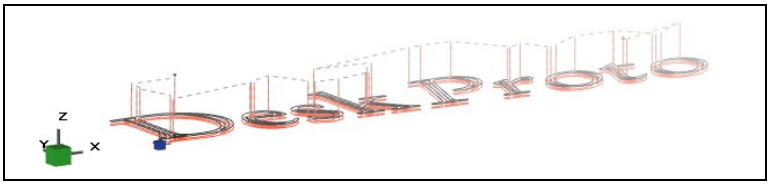
The parameters Name, Cutter, Feedrate and Spindle speed are completely identical to their counterparts in a 3D Operation. For the Cutter there is one important difference though: **in 2D toolpaths DeskProto does NOT compensate for the radius of the cutter !!!**

This means that the 2D lines as present in the 2D file and on the screen will just be converted to toolpaths without any cutter compensation. The resulting logo thus will be thicker than what you see on your screen, the difference being the radius of the cutter. This has been the reason to add “R0p5" (for inch users “R0p02") to the file name: the cutter radius for which these contours have been designed. This cutter compensation has been added in the 2D CAD program, using an offset command.
As this Sample file needs a flat cutter of 1 mm diameter, select this cutter.

A new parameter is the Machining level Z-value, defining the Z coordinate for the toolpath. You can best see the 2D toolpath as a **pen-plotter** operation: the pen (so here the cutter) will operate on two Z-levels. The pen will draw lines at pen-down level, and in-between it will make positioning moves at pen-up level. The same happens while 2D machining: the Machining level Z-value

defines the pen-down level, and the Free Movement height (on the third tab) defines the pen-up level.

Set the machining level to -0.5 mm (-0.02") and close the 2D Operation parameters dialog. Now calculate the toolpaths. The resulting toolpaths can be easily interpreted as they are identical to the contours but located half a mm lower. The positioning moves (a gray dashed line means a rapid movement) are on the default Free movement height (5 mm / 0.2").



You can write an NC program now, and engrave this contour into a flat piece of material. You will see the use of the "filling lines" inside the contour: when not present, islands of material would remain inside each character. This does not refer to say the large island inside the "D" as that is an intentional one, but to islands inside the thick vertical parts of the D. Using a thicker cutter would solve that, however would also make the result less detailed. An easy solution to create filling lines in CAD is to apply Hatching within the 2D contours. For most 2D CAD software hatching is a standard command to fill the inside of a closed contour with parallel lines.

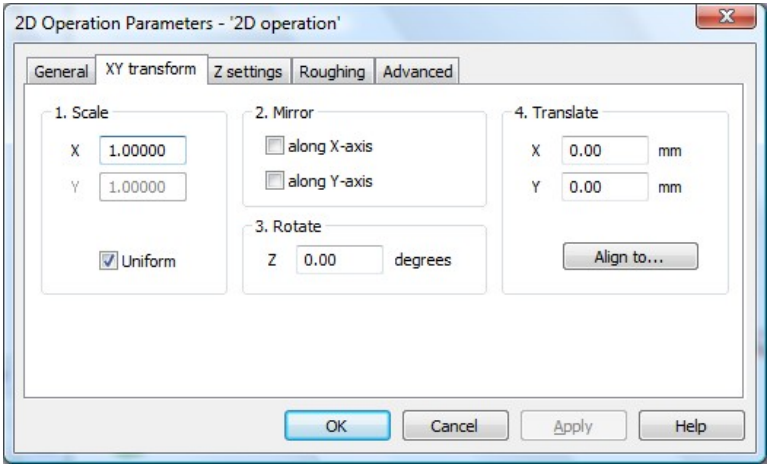
You will especially need to do so when you use a conical cutter with a very small tip: such cutter will give you the best (sharpest) resulting characters when engraving text.

Combining 2D contours with 3D geometry

As you have just seen, DeskProto's 2D machining option can be used to simply engrave 2D text on a flat surface. This is of course a nice application, however, in most cases you will use DeskProto for creating 3D parts, and use 2D for some extra detailing of this 3D part. A new issue then has to be taken care of: positioning the 2D contour data on the 3D geometry.

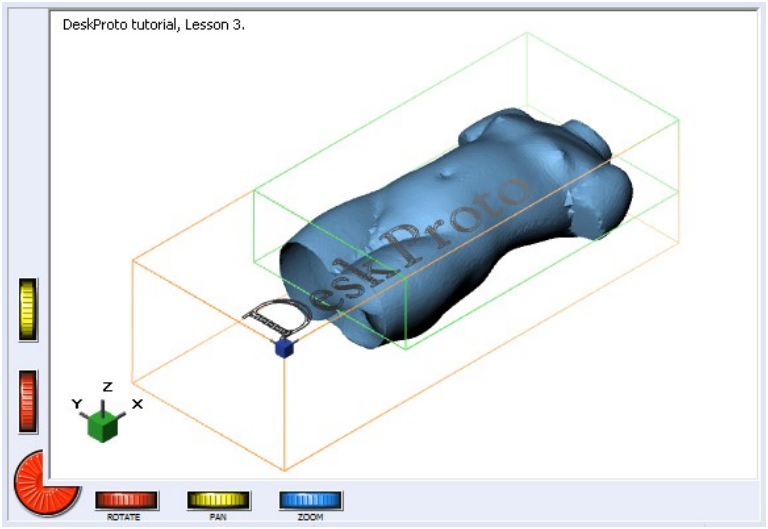
This positioning is an issue as DeskProto interprets the 2D file in **WorkPiece coordinates**, so in the coordinate system as used on the machine. 3D geometry files are imported in **CAD coordinates**, and after that transformed (rotated, mirrored etc) and translated. As some of these transformations do not make sense for 2D contour data they are not applied on 2D files. Also, in many cases the 2D file won't even be in the same coordinates as the 3D file, like a 2D logo to be projected on a 3D part, each created using a different design program.

In case you have created your 2D and 3D files with one CAD package, in the same coordinate system, you can automatically align them in DeskProto by not applying any 3D transformations (scale 1, no rotation etc) and by choosing None for 3D Translation on all three axes. Your will then see that both 2D and 3D files in DeskProto are positioned just as in your CAD software.



In any other case you can position the 2D contour data using the 2D Operation parameters on Tab page “XY Transform”, as shown above. Their use will be clear: Scale to change the 2D size, Rotate to rotate round Z and Translate to change the position in 3D space. For translating you can use the Align button to have the 2D contour aligned with some other object. The Apply button is very convenient in order to see what you are doing.

For this lesson we will use the Sample project TORSO.DPJ (or Torso_inch), and combine this small statue with the DeskProto logo and a few 2D lines. The idea is to engrave the DeskProto logo in the socle of the statue, so the 2D contours will have to be rotated, translated and scaled.
Open the Torso project, add a 2D Operation in Part Front, and in the 2D Operation parameters open the 2D File *2D_DeskProtoLogo_R0p5_filled.dxf*, the same file as used earlier in this lesson. You will see a picture as shown below.



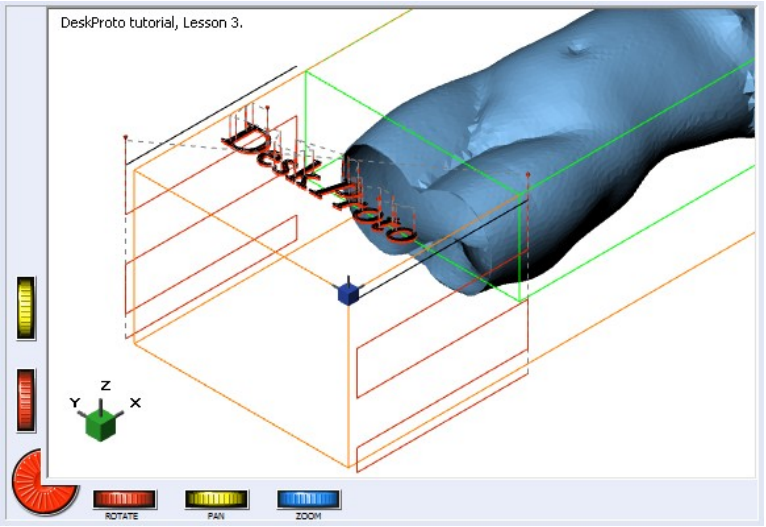
For a correct positioning of the logo on the Socle of the statue, the 2D contours will have to be rotated, translated and scaled. Open the 2D Operation parameters and use the Scale, Rotate and Translate options to position the 2D logo on the Socle to be created below the statue. By pressing Apply you can see the result without closing the dialog. The final result should look like the picture on the next page (we used Scale 0.5 / 0.5 , Rotation -90 and Translation 15 / 47).

Now set the machining level to minus 0.5 mm (-0.02 inch) which is sufficient for engraving purposes. The resulting toolpaths can be sent to the machine, and will match with the 3D toolpaths exactly as shown on the screen. So you can send both 2D and 3D toolpaths to the machine using the same workpiece zero point (do not forget to first mount the correct cutter !)

2D machining can be used for more than just engraving text. For this statue you can for instance use it to make a nice rectangular Socle. As the Socle has not been defined in the 3D geometry, your original block will remain at that location, needing some finishing. As an example for this lesson we have prepared the 2D file *2D_TorsoSockle_R2.dxf* (ballnose cutter D4 mm), for inch users file *2D_TorsoSockle_R1d16_inch.dxf* (ballnose cutter D 1/8 inch)

Create a second 2D Operation and browse this 2D file. Note that this file is an example only: which contours you can machine will depend on how you have fixtured the statue model. Translating these contours to the correct position is not needed: as you can see they are 2 mm (0.0625") outside of the socle, which is exactly the Radius of the cutter to be used. Set a machining level of minus

35 mm (-1.35") and select a Roughing layer height. The resulting screen will (roughly) look like the picture below.



In addition, 2D machining can also be used to drill positioning holes in your part, for instance for a correct positioning after turning upside-down (as an alternative to the ruler used in the DeskProto wizard). In order to have a hole drilled the 2D drawing needs to have a point drawn on that location. Using layers will result in “peck drilling”.

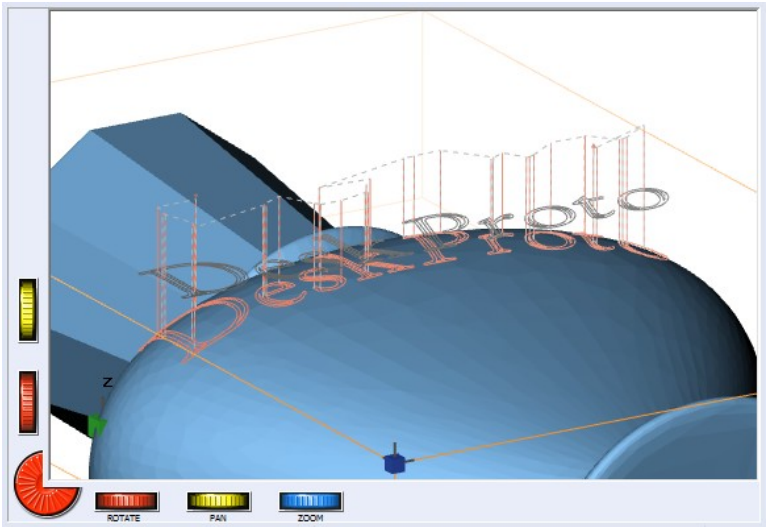
A nice detail is that 2D machining can also be used with rotation axis machining: the 2D drawing then will be wrapped around the 3D cylindrical segment, like a label around a jam-jar.

Projecting 2D contours on 3D geometry

One more 2D Operation parameter remains that has not yet been discussed: the option “Project 2D Contour on 3D Part geometry”. When you check this option the resulting Machining level (Z-value) will not be constant, but instead it will be applied relative to the Z of the part-geometry. The result will be that the 2D lines are indeed projected on the 3D part. Best to explain this using an example.

Start a new project
Load the geometry Bottle
Rotate the bottle minus 90 degrees around X

Set the part segment to Upper half only.
Add a 2D Operation
Load the 2D File DeskProtoLogo.dxf
Scale the logo to fit the bottle (we used 0.5)
Position the 2D logo over the widest part of the bottle (we used the “Align to” option to put the logo in the Center of the Part segment, both for X and Y).
Set the Machining level to minus 0.5 mm (minus 0.002 inch).
Check the option “Project 2D Contour on 3D Part geometry” and set the calculation precision for this projection to about 0.1 mm (0.0004”).
Calculate the toolpaths.



As a result your DeskProto screen will NOT look like the illustration above. As you have set the machining level to a negative value, the toolpaths are invisible as they are obscured by the rendered geometry. For the picture above we have set the machining level to +0.1 mm, in order to make a picture with visible toolpaths (these paths are of course not useful for machining)

Perhaps your toolpaths look ragged, and not as smooth as in the above picture. You can change that with the option Calculation precision, which is a sub-setting of Project 2D Contour. This parameter sets the calculation precision (grid size in XY) used to calculate the Z-value for each point on the 2D contour line. For a nice result a small value is best: for the example picture above we used 0.1 mm (0.004 inch).

The photo at the start of this Chapter shows the resulting machined prototype of the bottle with engraved logo. A nice result, with some extra remarks.

First: a good observer can see that a thicker cutter has been used than the R0.5 for which the 2D logo file was designed. Within certain limits it is possible to cheat with the tool radius (for the picture we used a ballnose cutter R 1.5 mm, and we kept the engraving depth very low).

Second remark: the projection used is vertical, which will distort the 2D logo: a circle projected on a tilted surface will become an oval.

On these tilted surfaces the logo may also be distorted by the cutter's thickness. DeskProto will project the 3D shape of the cutter on the 3D geometry, and when projecting on tilted surfaces the outside of the cutter will touch the geometry before the center of the cutter (which is on the 2D toolpath) does. The cutting depth is set from this first contact, so not from the point where the 2D toolpath is projected.
This last remark is valid only for DeskProto V6.1 (and newer): in older versions the center of the cutter was projected on the 3D geometry, not considering the actual cutter shape.



Lesson Six

Bitmap Machining



A second additional functionality of DeskProto is it's ability to machine 3D reliefs based on Bitmap data. DeskProto converts the 2D bitmap information to a 3D geometry (a relief), and then calculates toolpaths over this geometry.

The bitmap used for this example shows the Austrian Field marshal Radetsky (1766-1858, made famous by Johann Strauss). The bitmap file (grayscale image) may be used by courtesy of the HTL-Steyr, an engraving school in Austria. It has been made by 3D scanning an old relief made in plaster.

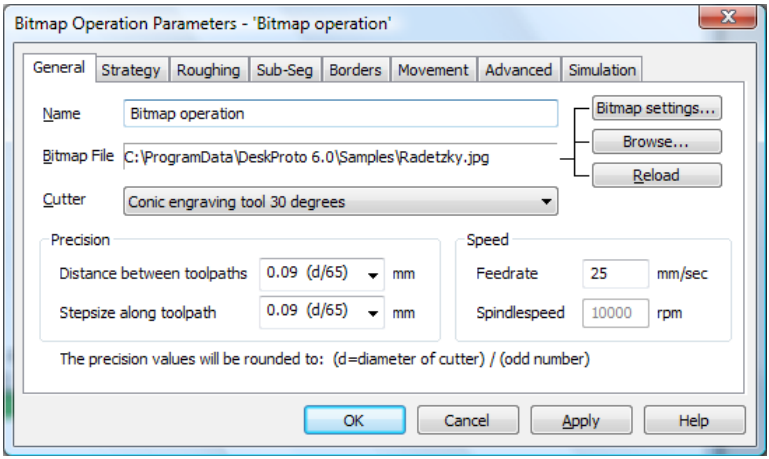
Creating a Bitmap Operation

Bitmap machining in DeskProto is done in a special type of operation: the Bitmap Operation. So the process works just as described in the previous lesson for 2D machining: you first need to create a Bitmap Operation, only then you will be able to import a bitmap file.

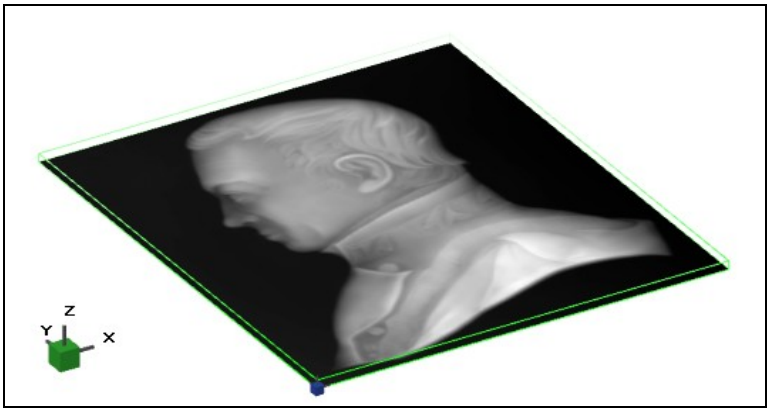
Again the easiest way to do this is to use the Start Screen, and select task “Start new project” >> “Bitmap project”. Make sure to first check the box “Use samples folder” so that you can easily open a Sample file later-on. A new line will appear in the tree (with a different icon) showing the new Bitmap Operation, and the default 3D Operation will be removed. The Start screen will also automatically open the Bitmap Operation parameters dialog for you.

Opening a Bitmap file

The dialog **Bitmap Operation Parameters** will look familiar, as most of the parameters are identical for Bitmap and for 3D. In case the dialog is not present you can open it by a double-click on its line in the project tree.



In your dialog the entry for **Bitmap File** will be “none”. Use the Browse button at the right to locate and open a suitable bitmap file. For this lesson we will use a great DeskProto Sample file called *Radetzky.jpg*. Without changing any other parameter press OK in order to see the Radetzky bitmap on your screen.



By rotating the view you can check that it is a pure 2D picture: one flat plane at constant Z-level. Only the color (gray value) of the pixels in the bitmap varies. For Bitmap files DeskProto supports BMP, GIF, JPG, PNG and TIF files. Color pictures are automatically converted to gray values (Black and White pictures) as for the conversion to Z-levels gray values are needed.

Converting 2D bitmap data to a 3D relief

The conversion is in fact very simple: each pixel has a gray value, which can be black, white or some in-between shade of gray. This gray value will be converted to a Z-value. You need to define Z-levels for black and for white, all in-between Z-levels will be calculated automatically. This is called
Gray-value to Z-height conversion.

You can imagine what happens if you apply this to the Radetzky picture. Give the black background the lowest Z-level (for instance Z=0) and assign a higher Z-level (for instance Z=5) to white. The result will be a 3D relief rising out of a flat background, the shoulder being the highest area as that is the lightest part of the bitmap.

One of the first ideas that may come into mind is to use a nice photo of your friend and convert that to a 3D relief. We have to disappoint you by explaining that the result will not be great. Imagine for instance a frontal picture of a face, with the sun shining from one side. One side of the nose will be light, the other side will be dark (shadow). The resulting relief will not resemble the original nose. Or image the difference between a white man with black hair and a black man with white hair. The resulting relief may be OK for your application, however do not expect it to be a copy of the real face.

Using a photo to create a 3D relief works very well for **Lithophanes**. This is a form of art where the relief is made in a thin sheet of material, and then illuminated from the back. The thinner the material, the more light will come through. The result will be that the original photo is clearly visible. For more information see the lithophane videos on www.deskproto.com

The Radetzky bitmap has been made using an existing relief and a 3D scanner, so has not been based on a plain photo. The shell bitmaps that you will find as examples also have been made using a 3D scanner, from real shells. When you can keep in mind how the gray-value to Z-height conversion is done, it is also possible to draw a picture in gray values for this application. Pictures like multicolor company logos can easily be converted to reliefs.

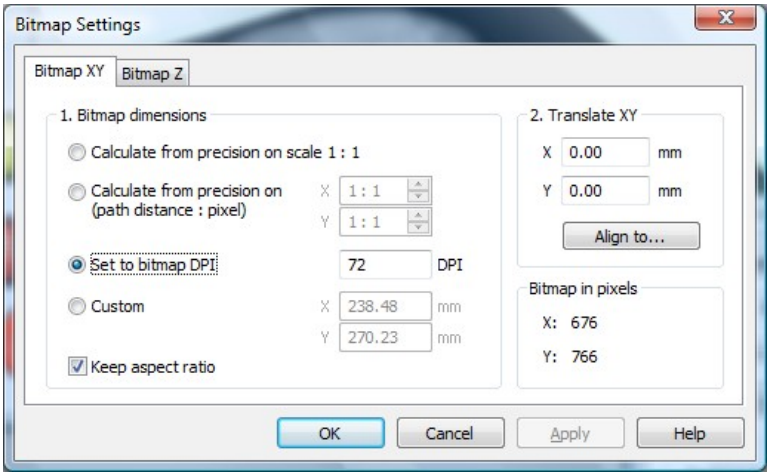
Bitmap Operation parameter settings

Now re-open the dialog "Bitmap Operation Parameters" that was just closed, and look at the available parameter settings. Most parameters are completely identical to their counterparts in a 3D Operation. Three buttons have been added to this dialog: the Browse button that you have just used, a Reload button, and the Bitmap Settings button.

First choose a **Cutter**. When you want to machine this relief on a size of about 50 mm (2 inches) then it is best to use a Conic Engraving cutter as on this size a small tip is needed to mill all the details. We have selected the "Conic engraving tool 30 degrees" for this project. This cutter has a tip diameter of 0.1 mm, so the Distance between toolpaths needs to be 0.1 or smaller. We have selected 0.09 for both **Distance** and **Stepsize** (inch users about 0.004 inch).

Extra **Bitmap settings** are needed as for a Bitmap Operation you need to define how you want to have the 3D geometry (the relief). For a 'normal' 3D Operation the 3D geometry is defined in the Part parameters, for the Bitmap relief it needs to be done at Operation level. Press the button "Bitmap Settings..." to open this dialog, where you can set the dimensions (both XY and relief height) and the position.

On the first Tab page "**Bitmap XY**" you can set the size and position of the relief in the XY plane. The bitmap can not be rotated in DeskProto: neither in the XY plane nor in 3D space.



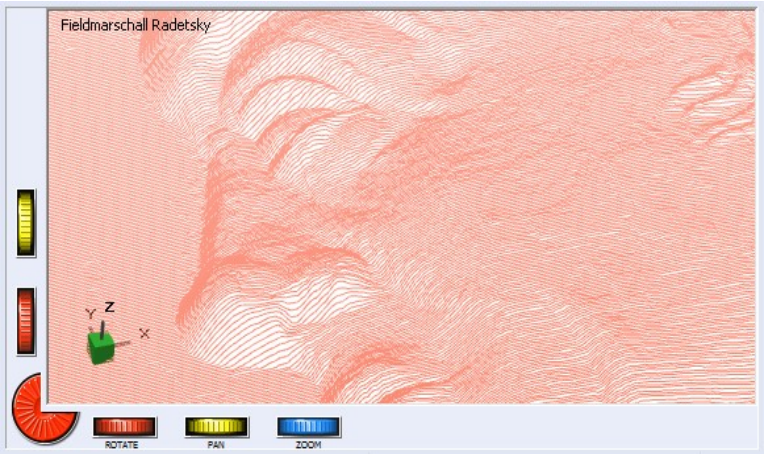
For the size (**Bitmap dimensions**) you can select one of four options. Options Bitmap DPI and Custom will be clear, just as the checkbox "Keep aspect ratio" If not then please use the Help button for more information. The two edit fields after Custom will show the resulting dimensions, for each of these four options.

The two options "Calculate from precision" need some explanation as these are not easy to understand.
In DeskProto toolpaths are calculated using the Z-grid: a rectangular grid of XY positions, with a Z-value calculated for each position. The Precision (Distance between paths and Stepsize along path) sets the size of each grid cell. This grid resembles the bitmap grid of pixels: each grid is a rectangular matrix along XY, with a Z-value/color-value for each position.

The option "Calculate from precision on scale 1 to 1" makes both grids equal. So the bitmap will be scaled to make one pixel exactly match with one cell in the Z-grid. In the second option a different ratio can be chosen to scale the pixels relative to the Z-Grid cells. For these two options take care: when you later change the precision this will automatically also change the size of the resulting relief.
Select "Calculate from precision on scale 1: 1", which will make the relief 62.40 x 70.71 mm (as you have set the precision on 0.09 mm)

Translate XY gives you the option to change the WorkPiece zero point. Normally this is at the left-front corner of the bitmap, however you can position it wherever you like. This will be needed when you combine a bitmap with a 3D geometry, for now you can just leave it on 0, 0.

On the second Tab page of the Bitmap settings, called "**Bitmap Z**" you can set the Z-values to be used for the relief. The meaning of the fields Z-value for White and Z-value for Black will be clear after the explanation in the previous paragraph. For the Radetzky relief you can choose 0 for black and 5 for white, of course depending on the size of the relief that you have just set. Inch users can set 0.0" and 0.2" respectively.
The relative Z-levels will be discussed later.



Now press OK for both Bitmap dialogs, and then calculate the toolpaths. The result should be something like shown above (after zooming in on the face).

You now can write the NC file and send it to you milling machine in order to mill the relief. The WorkPiece zero point is at the left-front corner of the relief, at the bottom of the relief (as you have set the relief between Z=0 and Z=5). So the top of the block needs to be on level Z=5.



Combining a bitmap with a 3D geometry

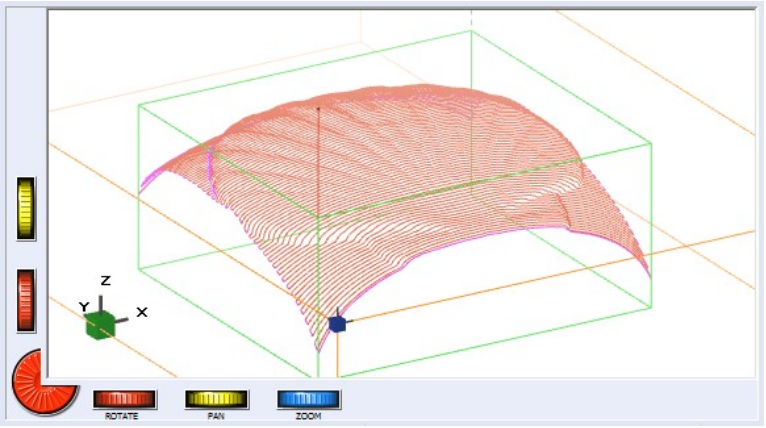
This subject will be covered only briefly, as most actions that are needed have already been explained in this Tutorial. The objective of this last example project is to create a perfume bottle (lessons 2 and 5) with a shell relief on the side of the bottle. Please do the following:

Start a New project
Load the Sample geometry Bottle
Rotate the bottle -90 degrees round X
Set the part segment to Upper half only
Add a Bitmap Operation
Browse the Sample bitmap file *Shell1.jpg*
Select cutter "Ball nose, radius 1 = diameter 2 mm" (a ballnose will smoothen surface irregularities in the relief)
Set Toolpath Distance and Stepsize on 0.12 mm.

Open the **Bitmap Settings**, tab Bitmap XY.
For **Bitmap dimensions** choose Custom and set the X-size to 34 mm. Y then will become 30.75, as the option "Keep aspect ratio" should be checked.
Play around with **Translate XY** and position the shell on the side of the bottle. This can best be done when you first display a Top View and then in the Bitmap operation parameters hit the Apply button after each change to see what you do. We have selected a translation of 10 mm for X and 20 mm for Y. Note that these values are correct for the default Translation in the (3D) Part Parameters (Positive X&Y for part geometry).

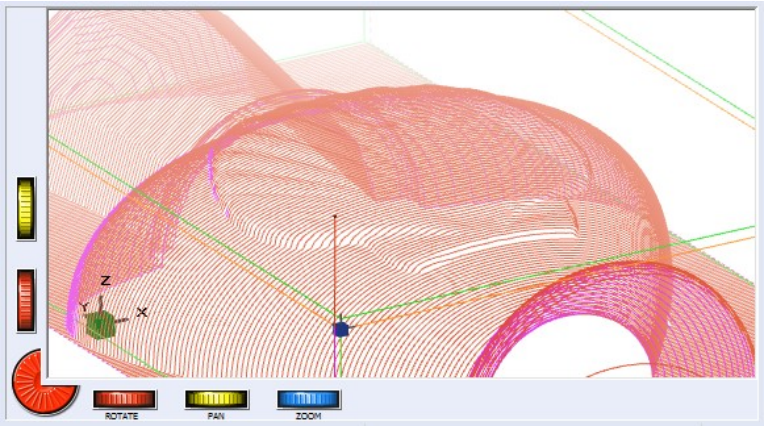
Now select tab Bitmap Z.
For **Z calculation** choose 1.0 for White and 0.0 for Black.
Finally check the option **Project bitmap texture on 3D part geometry**.

Press OK both for the Bitmap Settings and for the Bitmap Operation Parameters, en calculate the toolpaths for the Bitmap Operation. The result will be like the picture shown below. In the picture for clarity less toolpaths have been drawn, and subjects Geometry and Bitmap have been switched off. Also the 3D Operation in this project has been set to invisible.



As you can see the bitmap relief has been projected onto a curved surface: the bottle geometry. A nice positive relief: laying on top of the surface. We now are facing a problem though: the first operation of this project (the one that machines the bottle model) will remove all material needed for the relief. So the toolpaths of this second operation would only be cutting in air.
For a negative relief (with negative bitmap Z-values) to be engraved into the bottle this would not be a problem: then this file could be sent to the machine.

We can solve this problem by making all operations in this project bitmap operations: so roughing, finishing and if needed detailing all will be defined using Bitmap operations with the same bitmap settings.

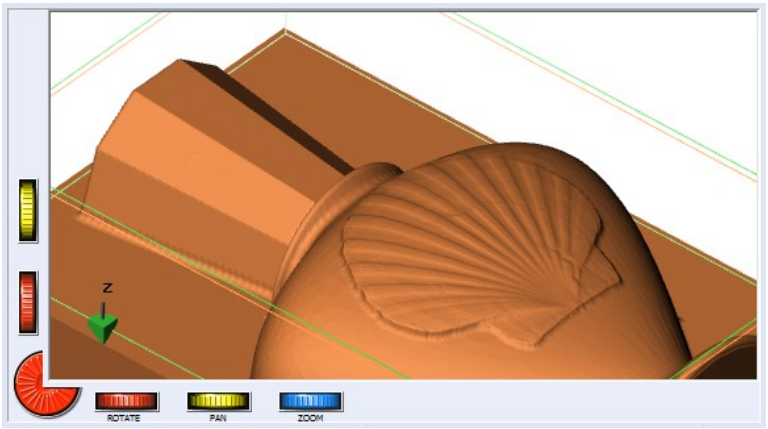


In order to make this Bitmap operation machine the complete part, two

changes are needed in the bitmap operation parameters:
- For Borders select **Extra for Cutter**. In the picture above you can already see that the outside of the relief was not machined as for bitmaps the default for Borders is No Extra.
- For the Operation segment select **Use part segment**.

After these two changes the toolpaths will look like in the picture above (again less toolpaths have been drawn for clarity of the printed picture). The extra toolpaths that you may see over the bottom surface are the result of using a multiple diameter cutter: it cannot reach the bottom of this vertical surface, as then the thicker shaft would collide with the top.

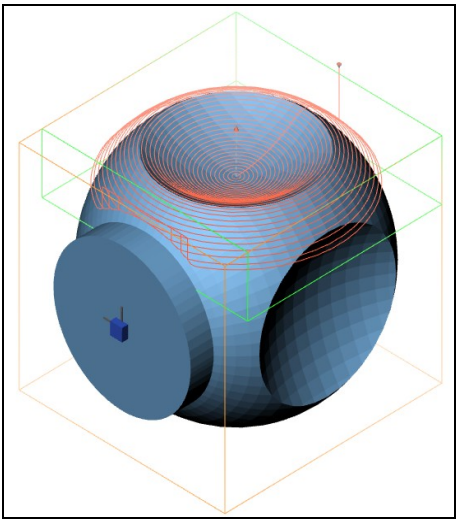
You now can delete the original 3D operation in this project, and copy the bitmap operation as many times as you need:
one to make it a Roughing operation, one for finishing, and if needed also one to detail out the relief with a much thinner cutter. For this last operation you then again can limit the Operation segment to “Use bitmap size”.



This last picture shows a simulation of the machined bottle.
Meant for a perfume that will surround you with the scent of a fresh sea breeze...

Lesson Seven

Five-Axis Machining



The Multi-Axis edition of DeskProto can generate toolpaths for Five-Axis machining. This is quite complicated matter though, and it involves much more work than using the fourth axis (described in Lesson 3).

For five-axis jobs DeskProto offers indexed machining: machine from one side using three-axis machining (X, Y and Z), then rotate A and B to a next orientation, again complete a three-axis operation, and so on. The number of orientations and their rotations may be freely chosen.

The geometry used for this lesson is a sort of "dented sphere" as shown above, you can find the file *DentedSphere.stl* in the DeskProto Samples folder.

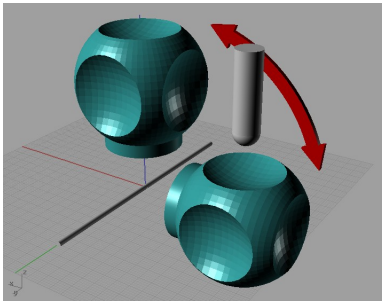
It will be clear that this lesson cannot be applied by users of the Entry edition and the Expert edition.

Five-Axis CNC milling machines

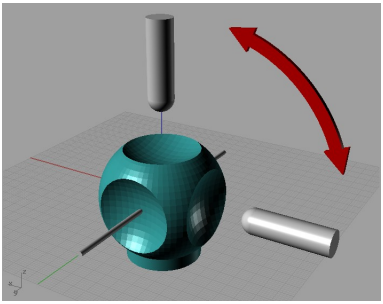
What makes five-axis machining difficult is the fact that so many different configurations of these 5 axes are possible. Machines are available for many of these configurations as well.

The theory is simple: a five-axis machine has three linear axes, called X, Y and Z, and also two of the three rotation axes, which are A (rotation around X), B (rotation round Y) and C (rotation round Z). So only three combinations are possible: XYZAB, XYZAC and XYZBC.

The rotations however can be applied in two ways: the cutting tool can rotate or the part can rotate. For a linear movement it does not matter whether the part moves or the cutter, for rotation it makes a big difference. You will also find machines that combine both methods.
And in addition the sequence matters: an X-axis built on top of an Y-axis will give the same results as Y on top of X, the results however for an A-axis built on top of a B-axis will be completely different from when B is built on A.



*90 degrees rotation of the **part**.*



*90 degrees rotation of the **cutter**.*

The DeskProto Five-Axis option is meant for **machines where the material block rotates and the cutter orientation remains vertical**. Only rotation axes A (rotation round X) and B (rotation round Y) are supported. Such machines are called **Trunnion-style** 5-axis machines.
It is also needed that the A and B rotation axes intersect, so have a common point: the center of rotation for both axes.

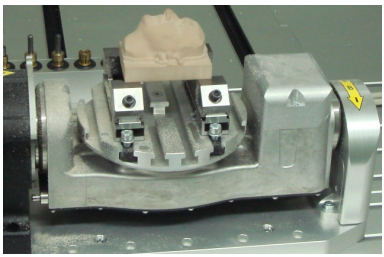
Rotation commands are used to make a different side of the geometry face upwards, and for that side plain XYZ machining is used. For each side in DeskProto a new Part is made, with a rotation set in the Part parameters (rotate round X and Y). The A and B rotation commands written in the NC file need to cause the same rotation on the machine.

Distance between rotation axis and geometry

A next detail to be considered is the location of the actual rotation axis (so the center of rotation). That is important as it determines the position of the geometry after the rotation.



5-Axis machine 1 (Roland DWX-50, for dental applications).



5-Axis machine 2 (Isel Euromod 45 with DSH-S rotation axis).

You can imagine the different behavior of the two machines shown above.
The machine on the left has it's rotation center in the center of the material block (a round disk of material, to machine dental crowns). So a rotation will not change the location of the center of this block.
The machine on the right has it's rotation center (far) below the block of material. It is even below the working table of the rotation axis (so the cutter may never go there). A 90 degree rotation of the horizontal axis will change the position of the block in the machine's working area.

It is of course very important that in DeskProto this same **center of rotation** is used for the part rotations.

This rotation centerpoint cannot be set in DeskProto though: DeskProto will always rotate around the three main axes (XYZ) as present in the STL file (so as set in the CAD system). The translation option cannot be applied here, as in DeskProto the translation is applied **after** the rotation.

The trick is to set the correct zero point in the STL file.
On the machine shown on the right, the distance between the point of rotation and the clamping plane of the machine vice (in the orientation as shown this is a vertical distance) is 76 mm.
In the file *DentedSphere.stl* the zero point is in the center of the round bottom surface.
So it is needed to translate this geometry over 76 mm along Z. You can do this in DeskProto (Part parameters, Translation tab, None for XY and Custom for Z), and then save the translated geometry using File > Save Geometry as > check to include the Translation.

In all cases it is important for the actual 5-axis project to set the **Translation** to **None** for all three axes (Part parameters). Otherwise the DeskProto translation will invalidate the positions that just had been carefully arranged.

As a result now the rotations done in DeskProto and done on the machine will have exactly the same effect.

Configuration of the two axes

Finally you need to check the configuration of the axes on your machine, and also the orientation that is present for a rotation angle of 0.0 degrees.
The result of course needs to match what you see in DeskProto.

DeskProto follows the ISO standard that the A-axis is parallel to (rotates around) X, and the B-axis rotates around Y. On some machines this is the other way round.

The rotations in the DeskProto Part parameters are done in the order XYZ: first the rotation round X, then the rotation round Y.
In case your machine features a B-axis built on top of the A-axis, then these same two rotation values can be used for A and for B rotation commands. This is for instance true for the Roland machine shown on the previous page.

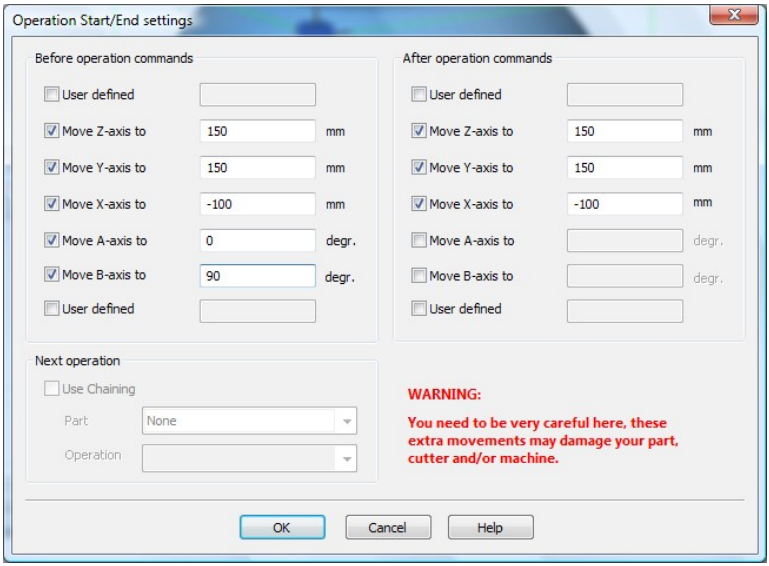
For other machines you may need to experiment to find the correct A and B rotation values to match the rotations round X and Y as set in the Part Parameters. This is for instance true for the Isel machine shown on the previous page, where a rotation of 0.0 degrees for the main rotation axis gives the second axis a vertical orientation. So this machine needs a 90 degree command on this main axis to match the 0.0 degree position in the DeskProto Part parameters.
For this machine also the naming of the axes is different: here the B-axis is parallel to X (normally this is called the A-axis). Of course this depends on how the rotation axis unit is mounted on the machine's working table.

Rotation direction of the two axes

Just as with 4-axis machining you will need to check the **Rotation direction** of your rotation axes: will they rotate clockwise or counterclockwise for positive values of A and B. We have found that no clear standard direction exists: each manufacturer makes his own choice. If your machine and DeskProto rotate in opposite directions you can correct this in the DeskProto postprocessor: Options > Library of postprocessors > OK on warning > select your post and press Edit > Tab Movement > add a minus sign to the value in edit box “Factor” in column A and/or column B.

Defining the A and B rotation commands

As said before, in DeskProto the rotations of the geometry are set in the Part parameters: transform tab, rotations round X and round Y.
In addition rotation commands need to be defined for the NC program file, to make the 5-axis machine perform the same rotations. This can be done in the **Start/End settings** for each Operation: Operation parameters, tab Advanced, button Settings for Start/End.
See the illustration below.



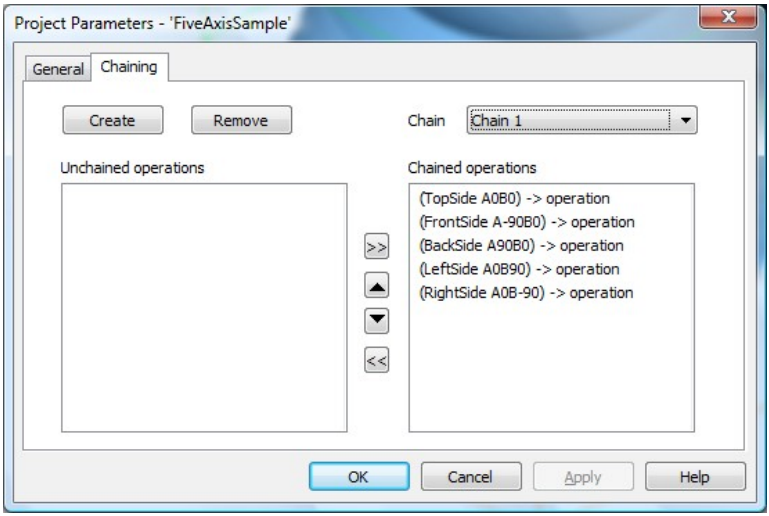
Here a series of commands can be defined to be added to the NC file at the start of this operation (the before commands), and a series to be added at the end (the after commands). In the illustration above, at the start of the operation the A-axis is moved to 0.0 degrees, and the B-axis to 90.00 degrees. At the end of the operation no rotations are set: that will be done in the next operation.

Very important is that values are entered for X, Y and Z, to send the cutter to a **safety retract position** before the rotations start. For many machines this is absolutely needed to prevent a collision between the rotation axis hardware and the cutter. Such collision will damage your part, and perhaps also the cutter and/or the machine. So take care !
The commands in this dialog are executed in the order as shown in the dialog: so first the Z-axis will moves (up), then Y and X will move, and finally the rotations will be done.

Movement commands for A and B can be entered only in case the machine definition used for this part indeed has these two axes defined (tab Advanced). If not then these fields will have been grayed out.

Chaining the operations

Finally all operations can be combined in one **Chain**, allowing the export of one combined NC program file. Chaining can be done in the Start/End settings dialog just shown, however it is easier to create a chain using the Chaining tab of the Project parameters, see the illustration below.



The illustration shows the situation for sample project *FiveAxisSample.dpj*
You can see that one chain is used, containing all (five) operations. For more information about the use of this dialog please use the Help button.

In this sample project you can also check all other settings:
- five parts, one for each side, with appropriate rotation values.
- the Start/End commands that have been used for each operation.
- the segments for all parts and all operations
- for the circular strategy the custom Center needed to be set for each operation

Be careful using the NC data generated by this project on your machine: as said for your machine the settings may need to be completely different.

Anyway: the result of your project will be one or more NC files that can be sent to your 5-axis machine. The file(s) will make the machine do the complete

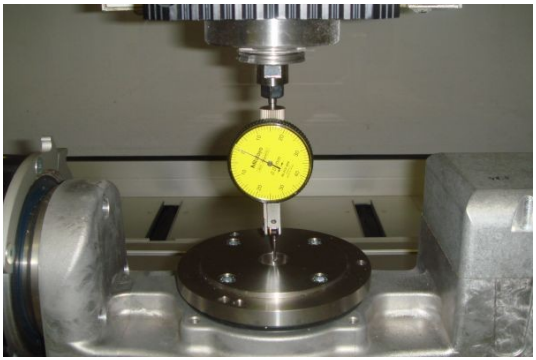
job: orient the part, machine one side, rotate to the next orientation, machine, and so on. You can of course also use Roughing and Finishing operations.
When more than one cutter is used, then for a machine with an Automatic Tool Changer (ATC) the result will be one NC file, for other machines several files will be written (new file after each toolchange).

DeskProto's Five Axis Sample project

The theory in the preceding paragraphs will now be illustrated by some pictures from the real world. These pictures were made by DeskProto user Robert Zeinecker from Germany, on his Isel Euromod 45 machine with Isel DSH-S rotation axis: thanks !

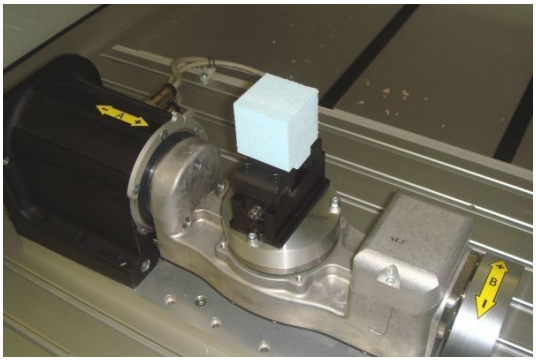
Robert adapted the sample project FiveAxisSample.dpj for his machine, and then machined this “dented sphere” in PolyStyrene foam. In fact he created his own version of this geometry by 'adding' a number of (1, 2, 3, 4 or 5) holes in each side, making it a large die.

Below we will illustrate the steps in this process.



The first step was to very accurately set the WorkPiece zero point: with the tip of the cutter on the exact spot where the A-axis and the B-axis intersect. This lesson does not show how this is done, some tips can be found in the documentation of the DeskProto 123WaxRing system.

On the rotation table a vice is used, and also the height of this vice was accurately measured: this is the distance between the zero point and the base plane of the geometry. This was 79.75 mm, so the geometry was translated to have it's base plane on Z=79.75 and then saved. Note that a small distance (between zero point and base plane) is preferred here: the larger the distance, the more any deviation in the zero point will be visible in the result.



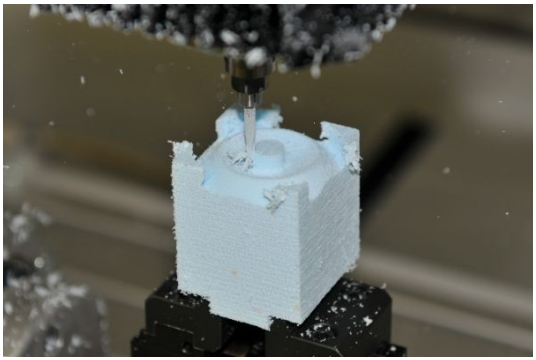
The configuration of the two rotation axes on this machine is not standard. The long axis is the B-axis, which on this machine is parallel to X. The round platform that can rotate is the A-axis: in the orientation as shown (B=0) it is parallel to Z (so should in fact be called C-axis).
A B-rotation of 90 degrees will make this axis parallel to Y. The machine then indeed has an A-axis and a B-axis, only these two have been switched (compared to DeskProto and to standard conventions).

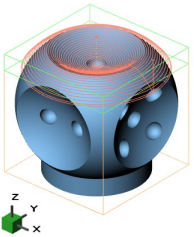
The photo above shows the first part (" Top"), the only part with the B-axis at 0 degrees. The blue cube is a PS foam block, clamped in a black machine vice.

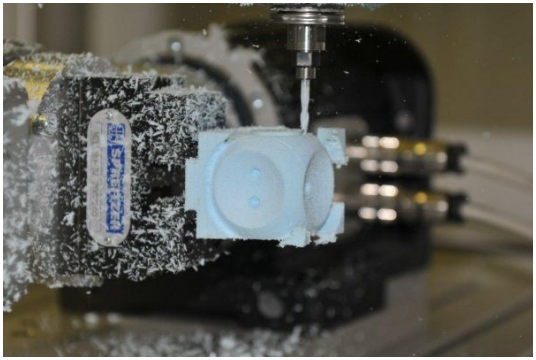
Some experiments were needed to find out which rotations in DeskProto matched which rotations on the machine. For this machine it was needed to reverse the rotation direction for both A and B, as you can see in the table below. As the two axes on the machine have been switched, the X part rotations in DeskProto relate to the B-rotations for the machine. The result was that the following five combinations of rotation values have been used:

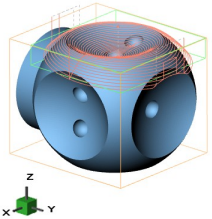
	in DeskProto	on machine:
Part Top	X 0.0 , Y 0.0	A 0.0, B 0.0
Part Front	X -90.0 , Y 0.0	A 0.0, B 90.0
Part Left	X -90.0 , Y 90.0	A -90.0, B 90.0
Part Back	X -90.0 , Y 180.0	A -180.0, B 90.0
Part Right	X -90.0 , Y 270.0	A -270.0, B 90.0

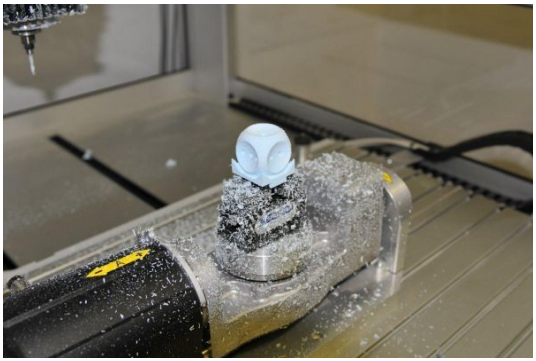
The rotation values for DeskProto have been set as Rotations in the **Part parameters** for the five parts, the values for the machine as "Move-to" **Start commands** in the Operation parameters. And here also movement commands for Z and for Y have been used to move the cutter to a **Safety retraction point** before rotating.
As said, the **Translation** has been set to **None** for all axes in all parts.
And finally **Chaining** has been used to connect all operations to one combined NC program file.

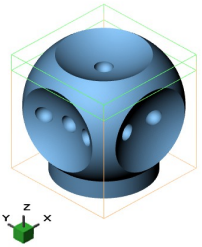












The pictures above show: the milling of sides one (Top), of side three (Left), and the result when all five sides have been completed. Look at the three green “orientator” cubes for the three axis directions in these pictures.

Note 1: Reversing the rotation direction can also be achieved in the postprocessor (Tab Movement, make field “Factor” negative for that axis).

Note 2: Creating a five-axis project is quite complicated and also error-prone (it is easy to forget one of the settings or make a typing error). It may be handy to create a **Template project** (project without geometry) for five-axis machining on your machine. That project should contain all rotation settings, start-commands, end-commands and Chaining, but no geometry and no segment settings. Then for five-axis machining you can open the template project, load a geometry and save the project under a new name.

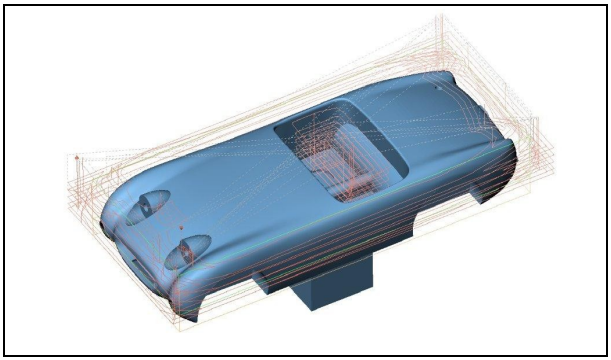
A Five Axis Sample car model

The cube project as described in the previous paragraph was just meant as a teaching help. Once you have mastered this project and have configured DeskProto to work with your five-axis machine, you can of course start with a real project.

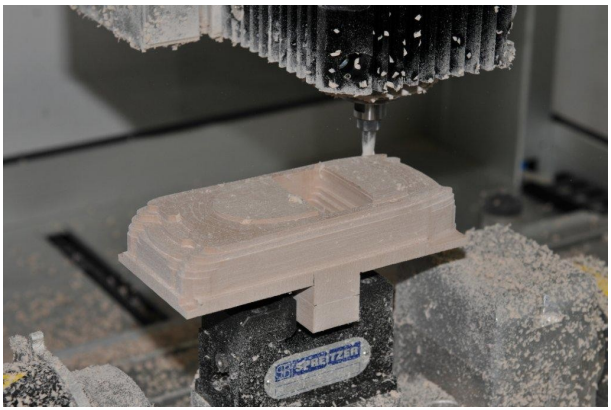
So did Robert Zeinecker, and he machined a great car model: an Austin Healey. Below you can see a few pictures: more can be found on the DeskProto Gallery at www.deskproto.com (section scale models). Including a video showing the actual machining process.



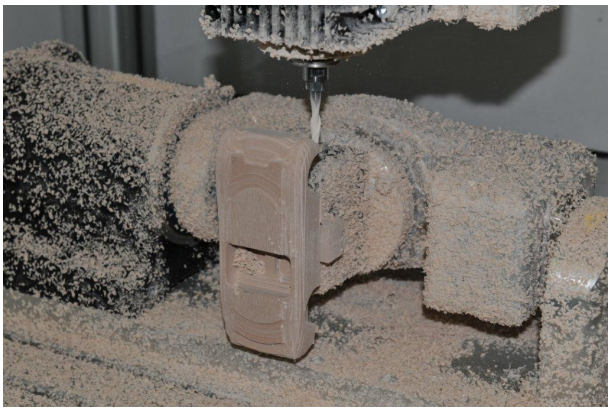
The car body geometry in CAD.

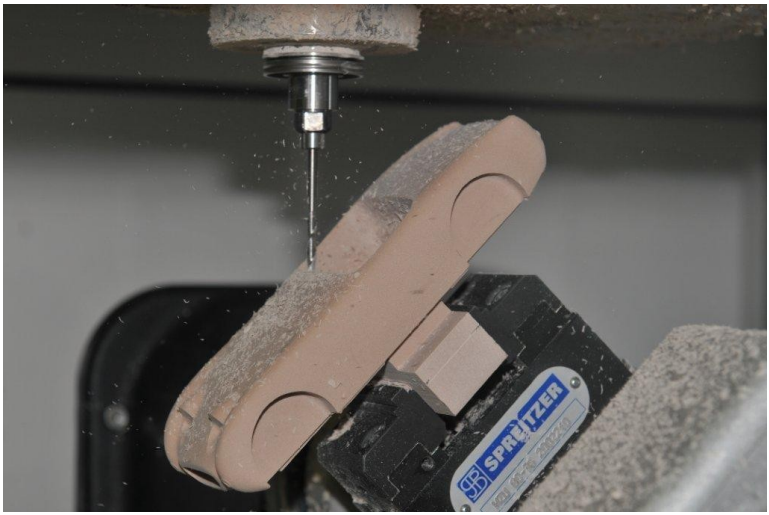


Roughing toolpaths for the top side, in DeskProto.

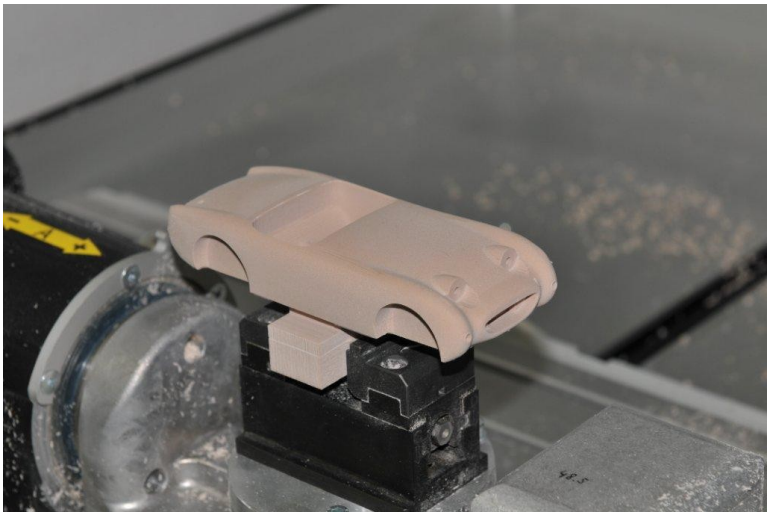


And the same toolpaths on the machine.





It is also possible to use angles other than 90 degrees, as shown above for detailing the dashboard of the car model.



The resulting model, after finishing all sides. Here indeed five-axis machining was needed to create this superb car model.

Index

2

2D File....77
2D Operation....76

3

3D relief....87

B

Basic 3D Wizard....19
Bitmap File....86
Bitmap Operation....86

C

Center geometry....56
Center of rotation....97
Chain....100
Clamps....36
Control software....37
Cutter....31, 45, 68

D

Dialog-based interface....17, 25
Dimensions....40
Distance between toolpaths....45
Drivers....13

E

Editions....17

F

Feedrate for high chiploads....49
File-locations....12
Finishing....34
Five-axis machining....96
FiveAxisSample.dpj....101
Fixturing....37, 58

G

Geometry Information....29, 40
Gray-value to Z-height conversion....87

I

Indexed machining....54, 95

L

Language....12
Layer height....35, 48

Lithophanes....88
M
Machine....10
Machining Time....46
Menu bar....15
Mouse Rotation....27
N
NC program file....33
O
OpenGL....9
Operation Parameters....45
Orientation....40
Orientator....26
P
Part Parameters....42
Postprocessor....10
Precision....31, 45
Project....17
Project bitmap on 3D geometry....91
Project Tree....17, 41
Projecting 2D contours on 3D geometry....82
R
Ramping....48
Register....10
Rotation....42
Rotation axis....54
Rotation direction....63, 98
Roughing....34, 47
S
Samples....13
Segment....43
Setup....9, 11
Skin....35, 48
Sort....46
Start Screen....20
Start/End settings....99
Status bar....15
Stepsize along toolpath....45
Subjects in View....28
Support blocks....57
T
Template project....104
Thumb-wheels....26
Title bar....15

Toolbar....15
Toolpaths....31
Trial Screen....20
Trunnion-style....96
Two-sided machining....66

U

Units....10

V

Views Layout....40

W

Wizard....16, 21, 67
Wizard interface....21
WorkPiece Zero point....36, 56