Desk*Proto*

Tutorial

Including Installation, Quick Start and six lessons.

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

Version 5.0

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

Contents

Disclaimer		5
Essentials		7
Installation		9
Quick Start		15
Lesson 1	The Casting (Basic milling)	17
Lesson 2	The Bottle (Two parts)	33
Lesson 3	The Venus Bust (Rotation axis milling)	47
Lesson 4	The Cellphone (Two sided milling wizard)	59
Lesson 5	2D Machining	69
Lesson 6	Bitmap Machining	79
Index		89

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 and any other intermediate parties are in no way responsible for any damage or injury, direct or consequential, relating to the use of this software.

Essentials

What does DeskProto offer

DeskProto is meant for any company that needs in-house facilities for Rapid Prototyping. Stereolithography machines and other layered manufacturing systems are far too expensive, however a light CNC milling machine combined with DeskProto is affordable, offering **Desk**top **Proto**typing. The advantages are clear: you no longer have to wait a number of days, your prototype is ready within hours! The design process will clearly be accelerated.
Do note that two different versions of DeskProto are available: **Full** and **Lite**, the latter offering a subset of DeskProto's functionality.

How does it work

The starting point for DeskProto is a 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 DXF files (in 2D Operations) and bitmap files (in Bitmap Operations).
Run the toolpath program on the desktop CNC milling machine in your own office, and you will have your prototype ready: *within a few hours* !

What hardware/software is needed

DeskProto is a MS Windows application (it needs Win 2000, XP, Vista 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 32 Mb RAM, 40 Mb free disk space and a USB port for the dongle: faster/more is better. An OpenGL compatible 3D graphics card is recommended

Installation

DeskProto version 5 runs with MS Windows 2000, XP, Vista or newer. Minimum required hardware is a Pentium PC with 32 Mb RAM: faster/more is better. A USB port is needed for the dongle (hardware lock to prevent illegal copies). The use of an OpenGL compatible 3D graphics card is recommended. For installation you need about 10 Mb of free disk space, to use DeskProto you need much more for the NC program files that you will create.

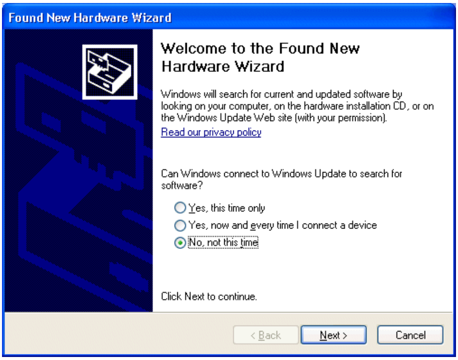
Note 1: Please **first** setup the software, and **after that** insert the dongle. Otherwise Windows might use incorrect dongle drivers.

Note 2: to install DeskProto, administrative privileges are needed (so you may have to ask the administrator to do the installation).

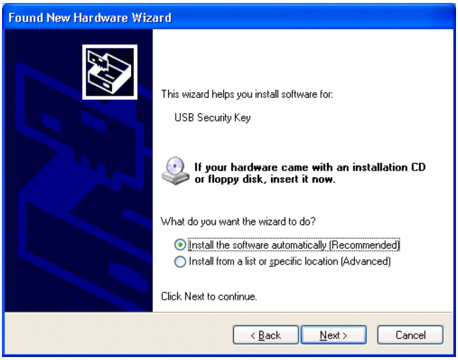
Insert the DeskProto CD in your CD Drive and the Install Menu will be automatically displayed : choose option 1 “Install DeskProto” to start Setup. Just follow the instructions given.
A new shortcut called DeskProto will be created on the desktop, and the necessary files will automatically be copied to your hard disk. Also the programs DeskProto, DeskProto-Help and Uninstall will be added to the programs list that can be accessed via the Start-button.

During Setup, Windows may ask you if it is OK to install software by Microcosm Ltd. You need to give permission as this concerns the drivers for the dongle used by DeskProto.

Now you can plug your dongle into the PC: the software won't start without ! After plugging in a USB dongle, Windows will start the "New Hardware found Wizard" (this may take a few seconds). As the Setup has just told Windows which dongle drivers to use, you can use the automatic procedure:



Do NOT let Windows connect to "Windows update", and



Let Windows install the software automatically.

Note: Please insure your dongle. Lost or stolen dongles will not be replaced. In case of a missing dongle you will have to buy a new license.

These dongle issues do not apply to users of DeskProto Lite.

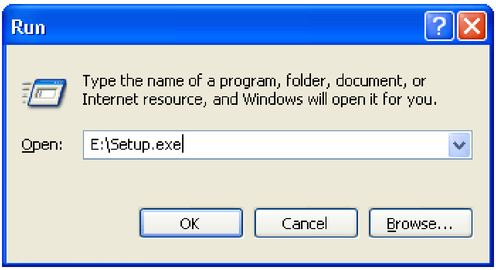
The first time that you start DeskProto it will complete the installation by asking you which CNC milling machine you will be using and which units (metric or inches). Do choose the correct machine (the one you have): it will be the default machine that DeskProto will use for all your projects.

Setup troubleshooting

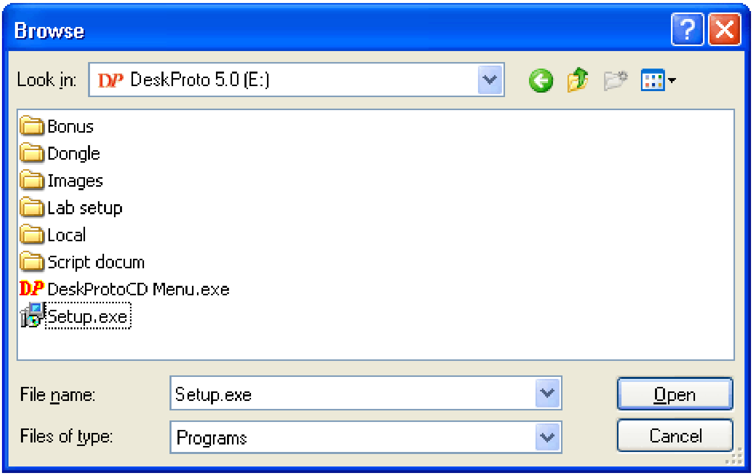
In case the DeskProto Install Menu does not show after inserting the CD, you can also start Setup using the Start-button in the lower left corner of the screen. DeskProto uses the standard installation procedure known from any other Windows application. The installation program is called SETUP.EXE, and can be started as any other Windows program.

Press the Start button and select Run from the Start menu that appears

Fill in the name of the program to be run in the Open field at the following dialog box, as shown on the next page. The drive name of the CD drive (E: in the illustration) may be different on your computer.



When you prefer this you can use the browse button: the following dialog box will then appear:



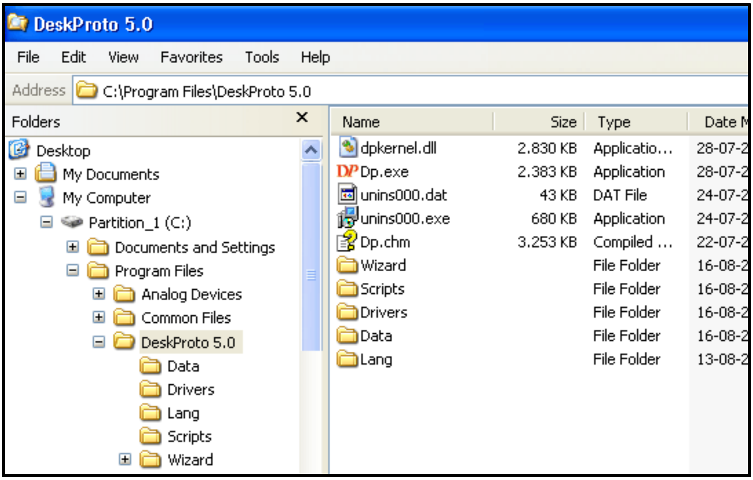
Select the **Setup.exe** program on drive D: or E: (CD) or, in case of downloaded files on your hard disk, and press the Open button. Do not mind the other programs that are shown. The Run program dialog box will then reappear, with the correct filename at the Open field.

After pressing the OK button the DeskProto installation program will be run.

Note 1: for Windows 2000 it may be needed to download the file GDIPLUS.DLL. This is needed in case the Setup returns error 1904 ("Module dpkernel.dll failed to register").
You can download gdiplus.dll from www.microsoft.com/downloads/
Copy this file to the same directory that contains DP.exe.

Note 2: in case of problems with the dongle or dongle drivers, please see the Advanced Issues section of the DeskProto Reference manual.

Files and directory-structure



By default, DeskProto stores its files in the following directories:

\Program Files\DeskProto 5.0
This directory contains the executables and the help-file of DeskProto.

\Program Files\DeskProto 5.0\Data
In this directory DeskProto by default stores all the data files: the Project files (*.DPJ), the Geometry files of the STereoLithography (*.STL), the Autodesk Data Interchange Format (*.DXF), and the Internet VRML filetypes. In this directory also the NC program files you produce will be saved, unless you specify a different location.

\Program Files\DeskProto 5.0\Drivers
For making an NC program that is suitable for your milling machine, DeskProto needs information about the correct machine, the available cutters and the postprocessor. This information is available from the configuration files (respectively *.MCH, *.CTR and *.PPR), stored in this drivers directory.

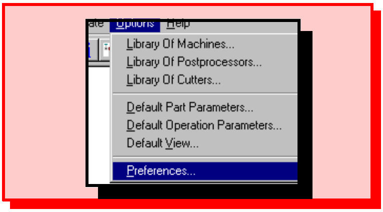
\Program Files\DeskProto 5.0\Lang
This directory (still empty after installation) is reserved for other **Lang**uage versions of DeskProto, and will contain translated resource files.
Each language needs two resource files copied into this directory, after which in the DeskProto Preferences the new language can be selected. The files are

called ResourceExeLang.dll and ResourceKernelLang.dll, where Lang is the language. For instance ResourceExeDeutsch.dll and ResourceKernelDeutsch.dll are needed for a German version of DeskProto. Note that also German Cutter-definition files and a German Help file are needed.

\Program Files\DeskProto 5.0\Scripts
This directory contains all Scripts, see the Reference manual (on your CD and on the DeskProto website) for more information.

\Program Files\DeskProto 5.0\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.

DeskProto will not influence any files outside these directories (unless of course you explicitly enter a different path specification for saving a file). As per general Windows requirements DeskProto will use the Registry to store all settings.



Note:
When the directory-structure has been changed, DeskProto must be reconfigured (choosing Options + Preferences) to be able to find the files again.

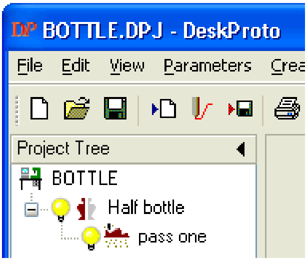
You can change the default directory location during the setup process: DeskProto then still will be able to find these files and subdirectories automatically.
However if you do change the places of the files afterwards DeskProto will no longer be able to find its files and might not run properly. In that case you can correct the DeskProto 'File Location' settings using 'Preferences' in the Options menu.

Quick Start

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

However, if your 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. After that, new users are advised to first use the DeskProto **Wizard**, that will guide them through all the steps needed to generate an NC toolpath file using their own geometry.

The central concept of DeskProto is a **Project**. 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).



Within one project 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. Each of these Parts has its own set of parameters:
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. Each Operation has its own set of parameters as well:
3. Operation parameters define how the geometry 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.

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. The basic Project consists of one Part called “Part” and one Operation called “Operation”. The project will be named when saving it for the first time. In the illustration the project is called Bottle, containing one Part called 'Half bottle' and one operation called 'pass one'.

Note: two different versions of DeskProto are available: **Lite** and **Full**. The Lite version only contains a subset of the available Part and Operation parameters. For the rest both versions are equal.

Note: to load a new STL file in DeskProto you must use NEW to start a new project. After that you have to choose "**Load Geometry**" in the File menu to import the STL file. You cannot use OPEN, as you do not yet have a DeskProto project file for this new project.
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.

The functions of DeskProto can be reached using the pull-down menus (or alternatively using the button-bar or the right mouse button). 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. In fact most of the functions in the View menu can be activated most easily by using the button bar.

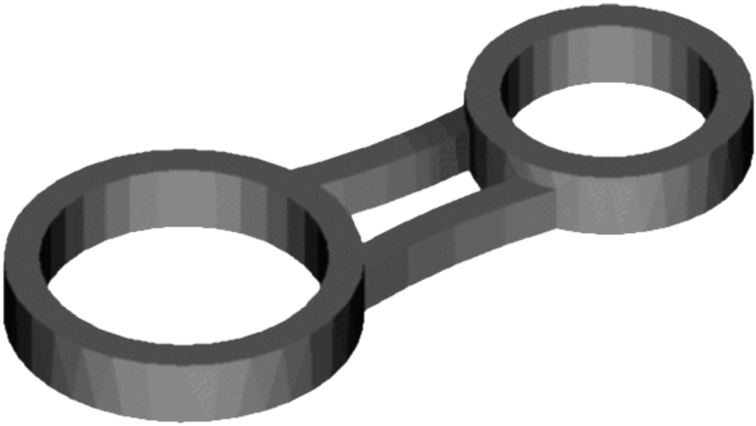
* In the **Parameters Menu** you can edit all geometry and milling parameters. For simple prototyping it is sufficient to edit only the front Tab screen for both Part and Operation parameters: the other Tabs can come later as all parameters have suitable default values.

* 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 1

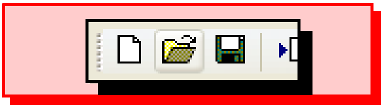
The Casting



In the first lesson the most elementary functions of DeskProto will be explained: you will learn what DeskProto can do and how to do it the standard way. A very simple object will be processed and an NC file will be made, ready to send to the milling machine. In addition to this lesson you can also use the **Basic 3D Milling Wizard** to teach you the DeskProto basics. This lesson is for both DeskProto Lite and Full.

The geometry is shown in the figure above: a very simple object, to be cast in metal. It was constructed (long time ago) in AutoCAD R13 using the solid modeling functionality, and exported as STL file. The prototype can be machined in one go (all geometry can be reached from one direction, without re-fixturing), which makes it a suitable example to start with.

Open an Existing Project

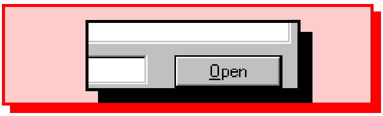


Select Open from the File menu or the appropriate button from the DeskProto button bar.

Every DeskProto session starts by creating a new project file or opening an existing one. In the project file all parameter settings are stored. To facilitate the lessons some predefined project files have been installed, so you can start by opening an existing project. Each action involves positioning the cursor on the correct position on the screen and then clicking the left mouse button.

Select the file CASTING.DPJ that has come with DeskProto as an example. It will be one of the files that are listed in the 'OPEN' dialog box.

The project file can be recognized by the extension DPJ. In the project file a reference to the geometry file, in this example CASTING.STL, is included, through which this file can be found and loaded automatically after opening the project.
Note: for users that work in inches special versions are available of the project files and geometry files needed for each lesson. Please select the file CASTING_INCH.DPJ

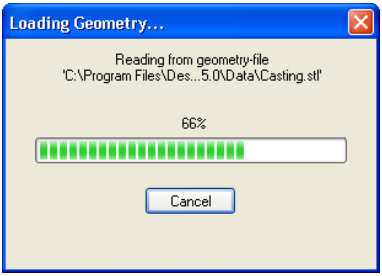


Press the Open button to start the Open function.

This use of this button and in fact of any OK button as well will not be repeated, from now on its use (when necessary) will be taken for granted.

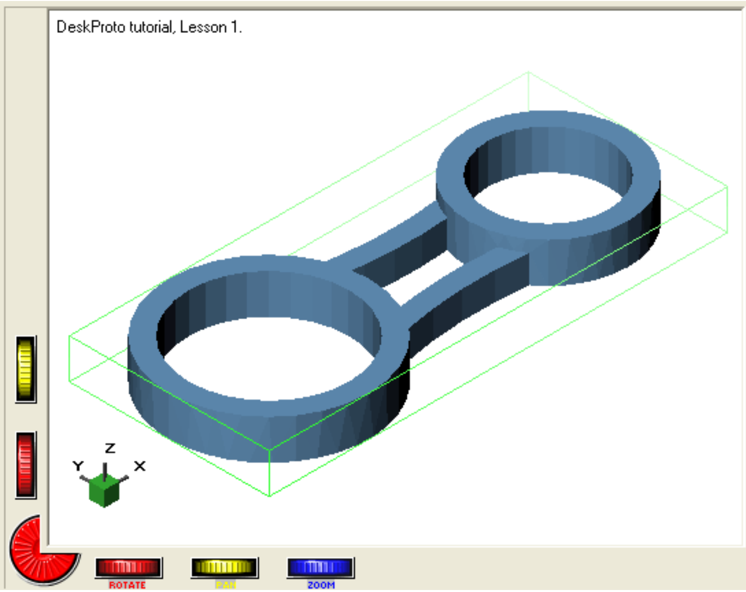
Load the Geometry file

When an existing project is opened, the corresponding geometry file will be read automatically. While reading the geometry file a progress indicator will be shown on screen, counting the percentage that has been processed:



The view window

When the geometry file has been read, the View window will show the geometry. In this example it is clearly visible (see the figure on the next page) that the geometry consists of ONLY small flat (in fact triangular) surfaces. These triangles or facets are the only type of geometry that DeskProto can work with, and are called **Polygon data**. Polygon data is the most commonly used geometry format for both visualization and rapid prototyping purposes. STL files always contain polygon data.
Note that in this example the triangles (facets) are very large: normally your geometry will have smaller facets and thus a smooth surface.

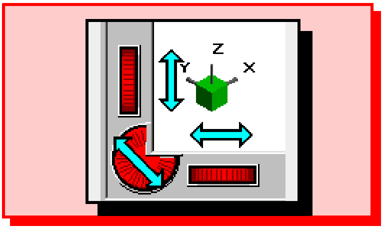


In this view, the geometry is being visualized by displaying the outer surfaces as solid faces of a given color. The orientation of each of the faces determines the shade value. This way of drawing is called rendering, and will be used by DeskProto as a default.

Rotate

DeskProto offers several ways of rotating the object, that is to view the geometry from any side. The rotation 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 corresponding rotations 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.



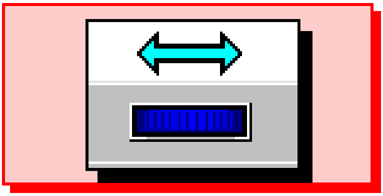
Move the mouse cursor on one of the red thumb-wheels at the left bottom corner of the window; press the left mouse button and keep it pressed while moving the mouse.
The round thumbwheel which is 3/4 visible controls the rotation around the axis perpendicular to the screen.

While these thumb-wheels are easy for rotations around one screen-axis, other controls offer an even more intuitive way of rotating. 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 (if not, check the mouse mode explained below).

A number of standard views can be set very quickly using the six 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 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 in space of your geometry.

Zoom

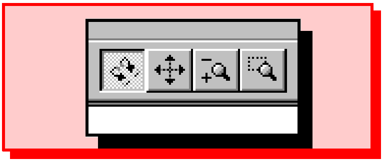
To take a good look at the model or some specific detail, in DeskProto you can easily zoom in on any part of the model.



Move the mouse cursor on the blue thumb-wheel at the bottom of the window; press the left mouse button and keep it pressed while moving the mouse to the left and to the right.

Watch the geometry grow and shrink as you turn the blue thumb-wheel by moving the mouse to the right and to the left.

For zoom as well, various controls are available. For mouse control you first have to switch from Rotation mode to Zoom mode. You can do so using the button 'mouse Zoom', which is the third of the four mouse-control buttons.



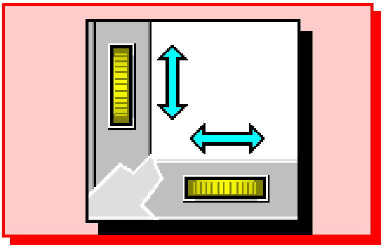
These four buttons define the function of the mouse inside the graphics screen. One button is always depressed, so one mode is always active. By default the left button is pressed for mouse rotation. Now press the third button and try the mouse zoom.

The fourth button is for mouse zoom as well, however now for Zoom Window. This is an option you will often use: use the mouse to draw a bounding box that defines the screen area to zoom into. This function does in fact influence both zoom and pan.

One more option is available for zooming: the mouse-wheel. Rotating the mouse-wheel makes the drawing larger or smaller. This option will work independent of which of the four mouse-mode buttons is active, which makes it very convenient. Note that the cursor position sets the center of the zoom, so you can zoom onto any detail on the screen.

Pan

The image of the geometry can also be moved over the screen (which is called panning), for example when after zooming you want to see a next part of the model. Therefore two yellow thumb-wheels are present at the bottom and left side of the window.

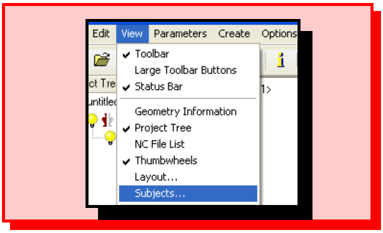


Move the mouse cursor onto the yellow thumb-wheels at the left bottom corner of the window; press the left mouse button and keep it pressed. Move the mouse to the left and to the right on the horizontal thumb-wheel, and up and down on the vertical thumb-wheel.

Using these thumb-wheels, the geometry can be moved inside the view window. The vertical wheel at the left side of the window controls the vertical displacement, while the horizontal wheel controls the corresponding horizontal displacement of the geometry. For an easier way of doing this switch to Mouse panning by pressing the second mouse control button.

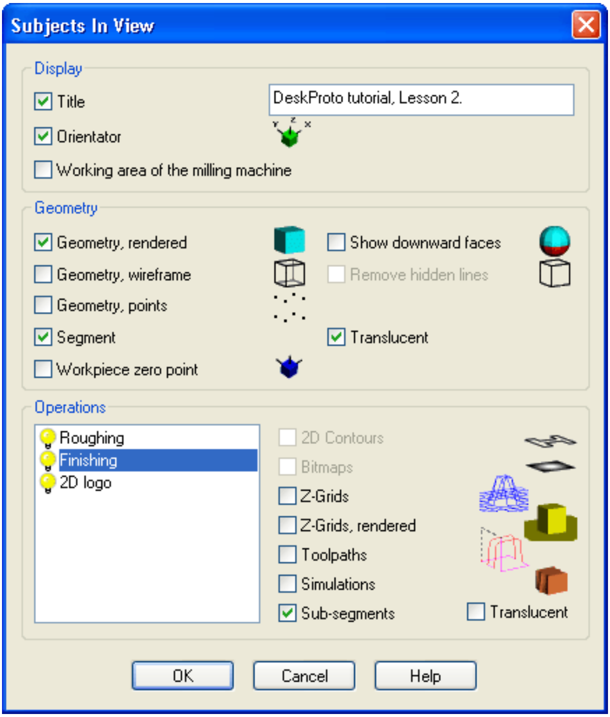
Also for panning one more option is present: pressing the middle mouse-button makes the mouse-movement pan your drawing. Note that the middle mouse-button is the wheel, which can be used as a button too. Again independent of the mouse-mode buttons on the screen.
So: *you can rotate, pan (press mouse wheel) and zoom (rotate mouse wheel) without using the mouse function buttons .*

Subjects in View

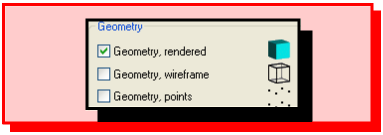


Select Subjects... from the View menu. You can also call this dialog by right-clicking inside the view window, by pressing the button "View Subjects", or just ***double-click*** *inside the View window.*

The dialog box that now appears controls which items will be displayed on the screen.

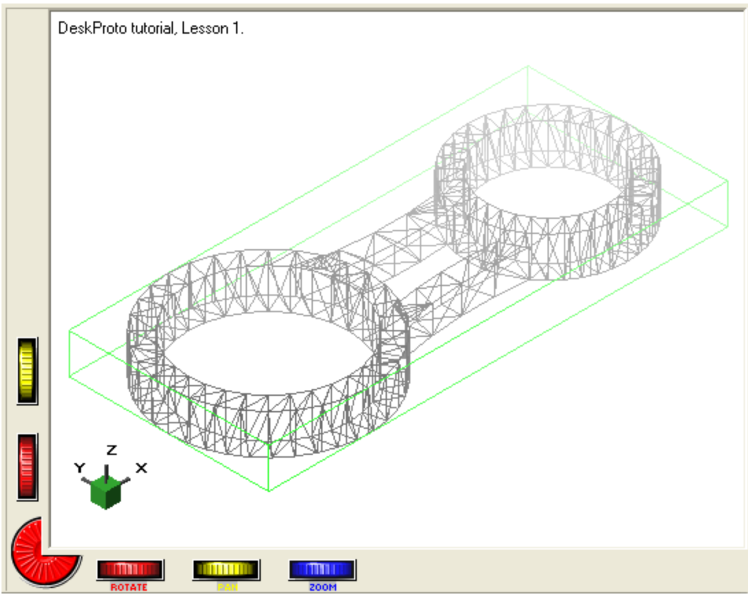


Every displayable subject is shown, with a checkbox to mark the subject to be displayed. These boxes can be ‘checked’ (made active) using the mouse. A checkmark resembling a "V" will appear in an active box, and the corresponding subject will be displayed after pressing the OK button. For now we will look at the geometry subjects only.



Check the “Geometry, wireframe” checkbox, deselect the “Geometry, rendered” box and press OK.

The rendering will be replaced by a line drawing (wireframe) of the casting, clearly showing the separate triangles of the STL file. On most OpenGL compatible graphics cards a 'fog effect' will be applied to enhance depth perception, see the illustration below. Also try checking and un-checking the two **Translucent** options.



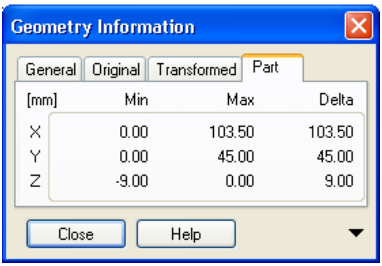
Check the model's orientation and dimensions

While examining the geometry by looking from some different directions, you may have noticed that the casting is positioned correctly 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. The small axes system with the colored cube drawn below left on the screen (called the orientator) is a very good help to check the orientation of the geometry. Note that the orientator only shows the directions of the three axes. It does **not** indicate the position of the WorkPiece zero point (0,0,0): that is available as a separate option in the Subjects dialog).

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

Press the Geometry Information button or select Geometry Info from the View Menu.

An information window will pop up telling you the dimensions of both the original geometry as present in the file (Tab 'Original') and the geometry of the model that you are creating (Tab 'Part'). More about the differences between these dimensions will be told in one of the next lessons.

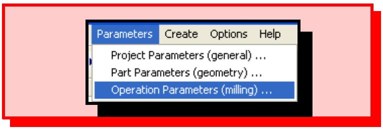


As you can see the dimensions of the part are OK (the illustration is in mm), so you can proceed and start creating your first prototype.

Check the milling parameters

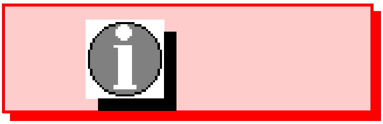
In order to calculate the correct NC toolpath (the path that the cutting tool will follow during the cutting process) DeskProto needs information about the milling parameters that you want to use. For instance the diameter of the cutter to be used, and how accurate you want to have your prototype. For this first lesson we will use the parameters that are already present in the example project Casting. (Nb: so make sure not to change these milling parameters, as someone else may be doing this lesson after you).

Still you need to check one thing: which tool will be used during the calculations. For the actual machining you need to use that tool. The use during machining of any other tool than this during calculations will result in an incorrect model.

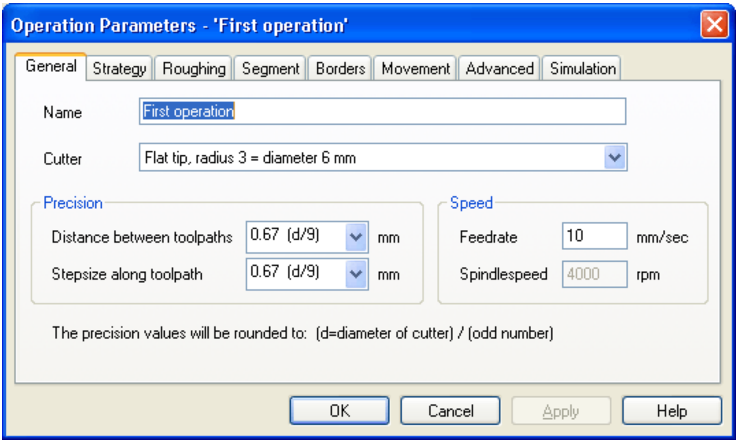


Select Edit Operation Parameters from the Project menu.
Note: *you can also* ***double-click*** *the line of this operation in the Tree.*

DeskProto will now display the dialog 'Edit Operation Parameters' which consists of a number of Tab screens. See the figure on the next page. 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.



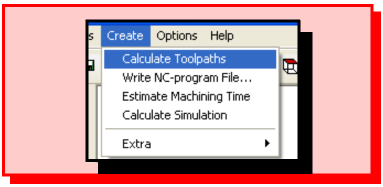
Try: Press the help button in the dialog box to get information about the parameters shown.



As you can see a tool with a flat tip (an end-mill) of 6 mm diameter has been chosen. For this geometry a flat tip is best: only horizontal and vertical surfaces are present. In case you do not have a 6 mm cutter you may choose a different tool. After pressing OK DeskProto will automatically check whether or not your machine can handle the specified feedrate and spindle speed. If not DeskProto will tell you, and you can easily change them to values that are permitted.

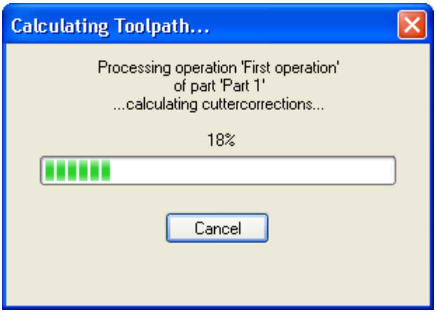
Calculate the Toolpath

After having checked the milling parameters you can now start the calculations.

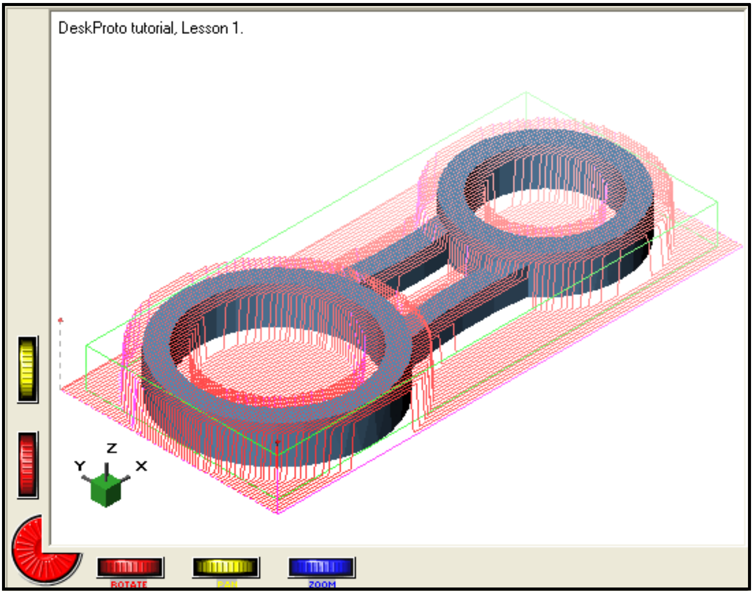


Select Calculate Toolpaths from the Create menu, or press the button "Calculate toolpaths".

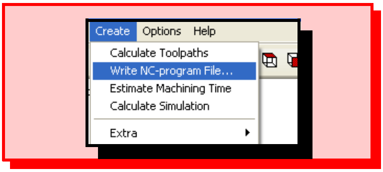
The process that will start now involves a number of subsequent calculations: first calculating Z-values, then adding corrections for the size of the cutter, and finally creating and optimizing the actual toolpath. During these calculations DeskProto shows a status bar to keep you informed about their progress.



After the calculations have finished DeskProto displays the toolpath: 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. Part of the toolpath is drawn using a dashed grey line: these are the positioning movements of the cutter above the prototype (ZFree level), which will be done faster (called Rapid) than the cutting movements (done using the Feedrate). The rising to ZFree after the last cutting movement is also done in Rapid (extreme left side of the screen).

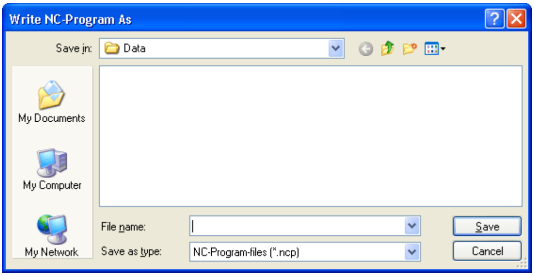


Create NC program



Select Write NC program from the Create menu, or press the button "Write NC file".

To send the toolpath just calculated to your milling machine you will have to first save it in a file, called the NC program file. 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. In the example below a machine has been selected that needs the file-extension .NCP.



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.

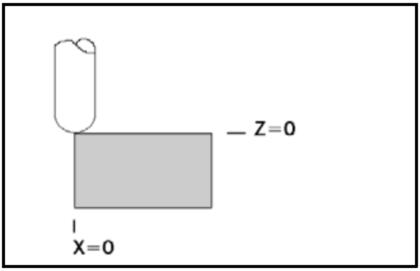
To the milling machine

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

First a block of material has to be prepared: you already know the dimensions. Make the block some millimeters larger for both X and Y, to compensate for possible positioning errors (not too large, for then the chips cannot easily fall off during machining). Make the block at least 20 mm larger in the Z-direction, to have enough room to fix the block on the milling table without risk of damaging your machine vise or milling machine. The easiest way to fixture the block is by using a machine vise. Alternatively for light materials like PolyURethane foam (PUR) double-sided adhesive tape can be used.
See Chapters 2 and 4 for more accurate fixturing.

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) 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), using its 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 the workpiece zero point, and in most cases also will be the starting point of the toolpath.



On many machines you can enter this 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 (default tool for the example casting project: a flat end tool of 6 mm diameter).

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 software to do this; exit DeskProto, start the 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 prototype will still be attached to the remaining block of material, as 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 bandsaw machine. This type of machine will by the way prove to be very useful in your prototyping workshop, to prepare blocks of material in the correct dimensions. In the next lesson you will learn how to create a prototype without any extra material remaining.

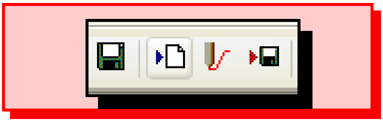
Lesson two
The Bottle

In this second lesson you will make a more thorough acquaintance with DeskProto. A prototype of a perfume bottle will be created, using a new project. You will thus have to enter the correct parameter settings in this lesson: no predefined settings are present as in lesson 1. The complete bottle will be milled in two halves, which means some extra care has to be taken when fixturing the block of material. This lesson is for both DeskProto Lite and Full.

The geometry was modeled in a CAD package called SIPSURF (no longer available), by Iris Timmers, a Dutch 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.

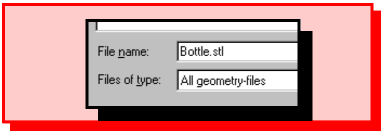
Start a New Project

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.



Select Load Geometry from the File menu or use the Load Geometry button from the DeskProto button bar.

The first thing to do now is to load the geometry file that contains the data of the model. Therefore DeskProto will ask for the name of the geometry file to be loaded, by displaying a File Open dialog box.



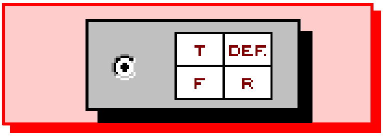
Select the file BOTTLE.STL that has come with DeskProto as an example (inch users choose Bottle_inch.stl).

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. 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.

Viewing the geometry

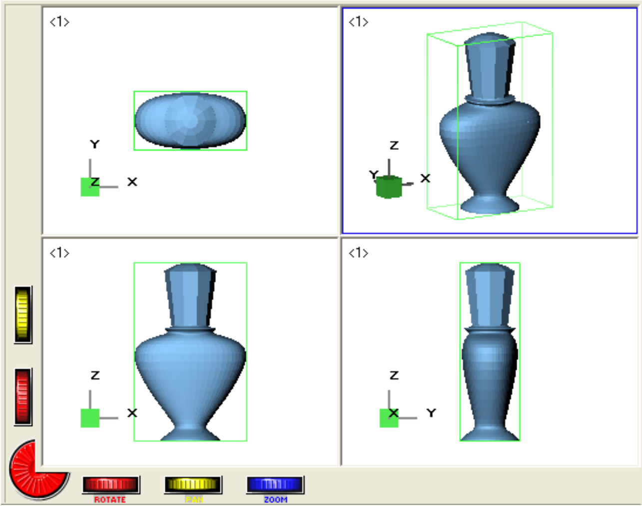
The first thing to do after loading the geometry file is to check the geometry. Make sure it is the right model, and check if its orientation, size etc are correct. To get the best view you can use a predefined standard Views Layout.

Press the Views layout button from the button bar, or select Layout from the View menu.



In the Views Layout dialog box choose layout 7, marked T/F/R/Def for Top/Front/Right/Default.

You will now get a window showing the three main views of the geometry and one “3D” view:

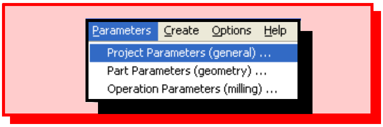


It can be easily seen now that the bottle is standing upright: its largest dimension is along the Z-axis. Since the cutting tool will come from the positive Z-direction, the model cannot be machined this way: you need to change the orientation.

The dimensions should be correct - to confirm this, check the button Geometry Information: a model of 86 x 54 x 28 mm (after rotation) does fit in your milling machine. The latter will be true for inch users as well.

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 prototype. 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**.

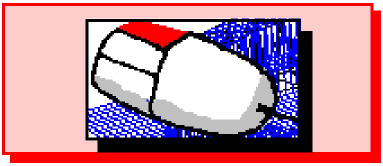


Select Edit Project Parameters from the Parameters menu

The Edit Project Parameters dialog box that you will see now does not contain many parameters. It contains the name of the geometry file and the names of all parts. **A Part contains all parameters that define the geometry to be milled** in one fixation of the material block. So for many projects (like in lesson one) one part is sufficient. For more complex prototypes more than one part must be milled: for this bottle you will mill a separate front part and back part. So in DeskProto two parts should be needed as well, however as for this bottle both parts are equal, and so one part is sufficient for the software.

As you can see your new project does not yet have a real name (it is called 'untitled'). You cannot enter a name here: it will be asked when saving the project for the first time. So no action has to be taken here: you can leave the dialog box using the Cancel button. 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.

Right mouse button

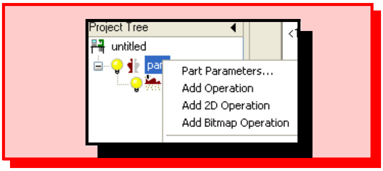


Click the right mouse button with the cursor pointing anywhere on the view

A user-interface related subject of this lesson is the use of the right mouse button. You can see now (do try !) that clicking the right mouse button either inside the graphics view or on the Project Tree brings up a 'Context menu', offering suitable options to change the settings of that view. You will soon realize that this is a very convenient way of using DeskProto.

Edit Part Parameters.

You can for instance change the Part Parameters by right-clicking on its line in the tree. The context menu will now offer the possibility to open the Part parameters dialog

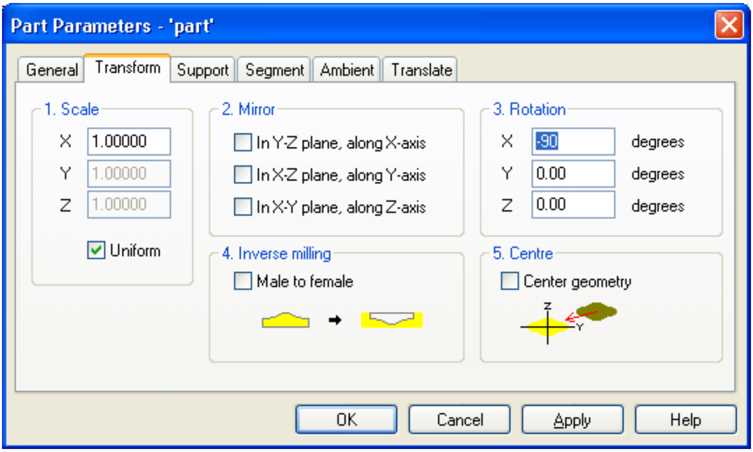


Click the right mouse button on the second line of the project tree (the line showing 'part'), and then click 'Part Parameters'

Much more convenient is though to just **double-click the line in the Tree**.

The dialog box 'Edit Part Parameters' will appear, presenting a number of Tab screens. **The 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 positioned. You will do so now: rotate the bottle to achieve the correct orientation for milling.

The first Tab screen is not very interesting. 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.

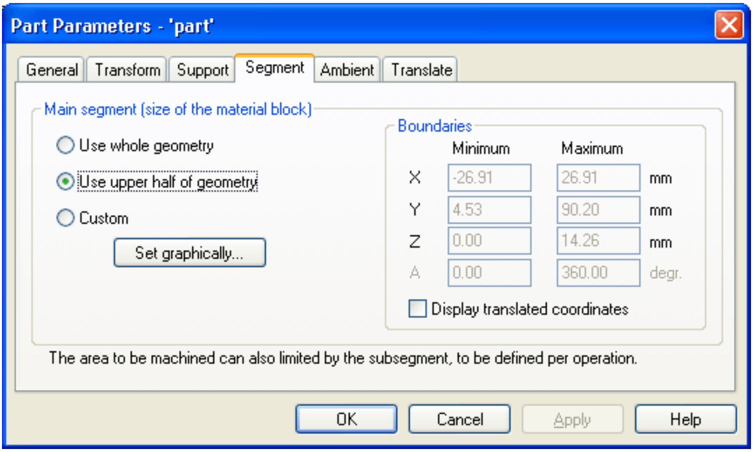


For this bottle model we need to set parameters both on the Transform and on the Segment Tab. The Transform Tab shown above makes it possible to change the size and orientation of the geometry: in many cases you do not want to use the exact contents of the CAD file. For instance in case of an automobile you may want to make a scaled-down model. Or in case of this bottle that is 'standing upright' it must be laid flat on the table to make milling possible. In this lesson enter a rotation of -90 degrees around the X-axis. Since for most milling machines the X-axis is the longest, a rotation of 90 degrees round the Z-axis may be useful as well. Enter the required rotation value(s) and see what happens after pressing OK. The orientation of the geometry should be correct now.

The other parameters on the Transform screen 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 prototype that is created. You can see on the screen that the geometry rotates while the XYZ cube-of-axes (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 prototype, 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. The geometry of the complete bottle is present in the file and on your screen. However this complete geometry cannot be milled in one part using a three axis milling machine. The prototype has to be milled in two halves, so toolpaths have to be calculated for half a bottle only: the upper half. This means a segmentation in the Z-direction. Choose the Segment Tab.
Note: for DeskProto **Lite** segments are not possible: instead use the parameter “Bottom level” in the tab page Transform.

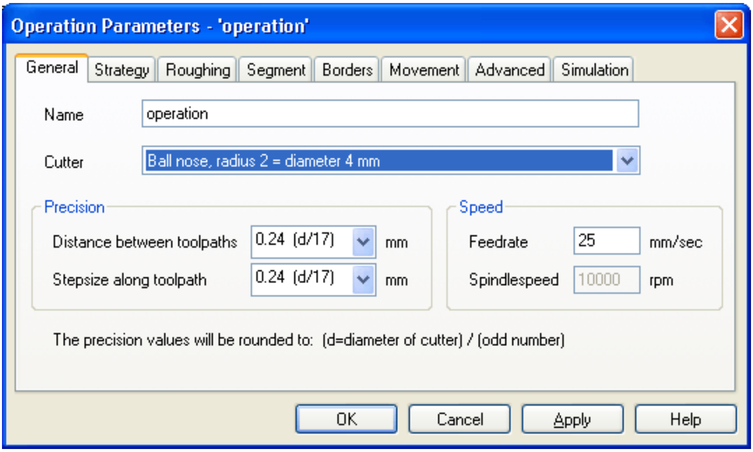
Segmentation means to use only a part of the geometry, being defined by a rectangular area (a 'Bounding Box'). By default the segment includes all geometry. We want to machine half a bottle now, and in fact exactly want to halve the bottle in Z. This is easy as it is a predefined option: just select option 2 (Use upper half) now. You can see that the min and the max values for Z were equal except for the minus sign, and that after choosing option 2 the Minimum Z of -14.26 has been replaced by 0. Press OK to exit this dialog.

In the drawing the segmentation limits will be visible as a green rectangular block. Note that all values for the segmentation are given in the transformed coordinate system, not in the original geometry coordinates. So the coordinates are after “Transform” (previous tab) and before “Translate (subsequent tab). In

some cases it will prove to be easier to enter the limits in translated values (as used on the machine), by checking this box.
Still easier of course is setting a custom segment using the mouse, which is possible after using the button “Set graphically ...”.

Edit Operation Parameters.

The Edit Operation Parameters dialog box 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), as just explained for the Part.



You will have to choose the correct cutter now. 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 best choice for those using inches would be the 1/8" ball nose cutter, 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 the available cutting tools use the option Library of Cutters in the Options menu. This library is where you can also define a new cutter, or modify an existing one to match your real tool.

The most important parameters here are the Precision parameters. They determine the accuracy of the prototype, and also the time needed for both calculating and machining. For a first rough prototype of the bottle a Distance between the toolpaths (also called stepover) of about 1 mm (0.04") is OK, for a final mockup a smaller value (like 0.3 mm) needs to be used. For a nice bottle we suggest to use 0.24 mm (= D / 17), for a fast model 0.80 (D / 5)

DeskProto offers predefined values for the Precision parameters. These are dependant 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".

The meaning of the **Distance between Toolpaths** parameter will be clear, this is also called the Stepover. As the path consists of many small straight line segments a second parameter is needed: the **Stepsize** (the length of these line segments). It is recommended to enter equal values for the Distance between toolpaths and the Stepsize, though for special cases you can experiment with different values for these parameters. Be careful with the highest possible value for Step size: D/1. When taking such large steps DeskProto may remove to much material !

A rough estimation of the machining time for your current settings is calculated 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 calibrating it for a better accuracy.

The default spindle speed (rotation speed of the tool in rpm) and Feed (traveling speed of the cutter) will be correct too. 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

As in lesson one, you can now start the milling calculations by choosing Calculate Toolpaths in the Create menu. Alternatives are: the button Calculate Toolpaths, the option Write NC Program (which detects that the toolpath has to be calculated first), and the Subjects in View dialog box where you can just make the Toolpath active in order to start the calculations.

You will see that (in case you did use cutter Ballnose R2) two different layers of cutter movements are calculated: the first at level Z = -15 mm, the second at the final depth. The reason for this is the cutting length of the 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.

Note: attentive readers will have noticed that the height of half a bottle is in fact only 14.26 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 (here 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 was not applied. 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 either now save the NC program file and continue with paragraph “To the milling machine” or add an extra Roughing operation first.

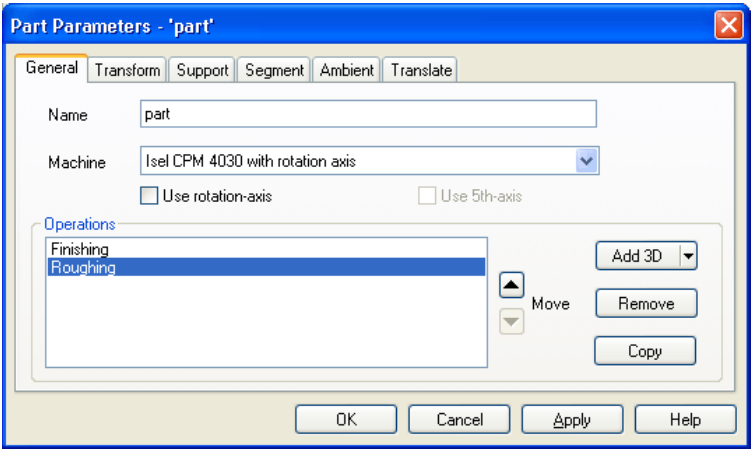
Optionally add a Roughing Operation

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 roughing 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 definition. 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 need not change cutters then 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.

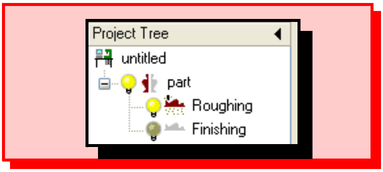
The new Operation’s line is automatically set in edit mode, so you can change it’s name from “operation [#1]” to “Roughing”. If this did not work then right-click on the operations line and choose Rename. The same way you can Rename “operation” to “finishing”. These names are not used in the NC file,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



Now only 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 on operation first. See the illustration above.
Note that after this change the toolpaths for Operation Finishing will be automatically recalculated, and the layer will disappear. As said before the layering had only been applied because it was the first operation for this part: now it is the second.

Next you can set the Roughing operation’s parameters. Open the Operation Parameters dialog for Roughing. First you need to select the correct cutter here as well: 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 remove during finishing) at 0.5 mm. Use the help button for more information. The Layer thickness can be fine-tuned: select Custom and enter a value for the layer height (thickness). For light materials like foam or tooling board you can enter 9 mm to have two layers that are about equal. For stronger materials like perspex or metal you will need to use a smaller value.

Do not forget to also choose new Precision values (tab General): both Toolpath distance and Stepsize can be set to 1 mm, in order to quickly remove the material.
Here again attentive reader may need some in-depth information: the dialog mentions D/5 for the 1 mm value, while the cutter has a diameter of 4 mm. What happens is that after setting a Skin DeskProto calculates 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.



Click on the light bulb of an Operation line in the Tree to make it invisible (light switched off) or again visible.

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 lines. It is easy to make it less confusing: in the Project Tree you can see a yellow light bulb on each line. Clicking the light bulb for an Operation will make it grey (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.
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 far less positioning moves are needed.
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 extra Ambient”. You will be able to see the difference immediately after pressing OK.
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

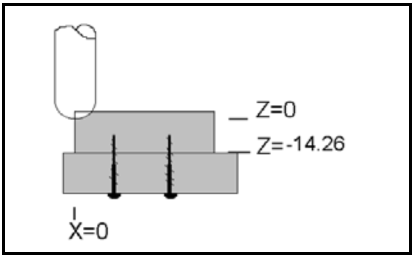
the feedrate to the percentage give here. For roughing in tooling board you can set this to 30 to 50%, for materials like perspex or aluminum even lower.

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).

To the milling machine

After writing the NC program file you can send it to the machine to create half a bottle. In lesson one you have learned how to fix the block of material and how to enter the (0,0,0) position. For this bottle prototype a different approach is needed, in order to get two halves that can easily be glued together for 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 for gluing. The problem is that now the block of material cannot be easily fixtured to the machine: on all sides of the prototype the milling tool needs operation space, the tip of the tool will even come **below the bottom of the block** and might damage your machine's working table. The solution is to screw two blocks together: the upper block to create the prototype, and the lower block to fix it on the machine. See the illustration below. The bottom block can also be a larger slab or piece of board, that you can bolt down to the machine with clamps. This way the tool movement area does not come near the clamps, which is of course very safe.
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:

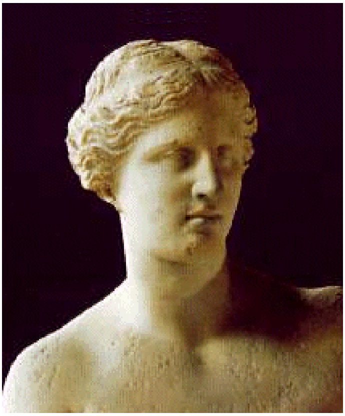
- The lower block has its top plane and bottom plane exactly parallel.
- The top block has a really flat bottom surface (needed for the gluing afterwards).
- The screw tips in the upper block are well within the portion of the prototype that will remain after all milling is done. Otherwise either the 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 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 (the symmetry plane of the bottle). Position the tip of the tool on the same height as the bottom of the block. Go up 14.26 mm and set the Z=0 level there. You can read this value 14.26 from in the DeskProto Geometry Information dialog box: Tab 'Part' shows that the minimum Z of the prototype is -14.26 (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, 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. Attach both halves together and your prototype is ready !

Lesson three
The Venus Bust

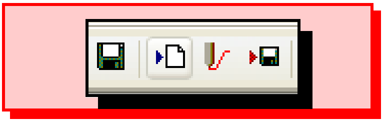


In the third lesson you will learn how to create toolpaths for a rotation axis: an extra item on your CNC milling machine that lets the object rotate during machining. It is also called A-axis, and looks like a spit on your barbecue. This lesson of course only is useful in case you have a machine with such an axis. Also: rotation axis machining is not available in DeskProto Lite.

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. As it is a bit larger STL file (7 Mb) the file venus.stl is not included in the standard DeskProto setup. You will first have to get the file from the Bonus geometry section of the DeskProto CD or download it from the DeskProto website, and then copy it to your DeskProto data directory on disk.

Start a New Project

This third lesson will again start with creating a new project, so that you can exercise 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 also use the Rotation axis milling Wizard to show you all steps to be taken.



Select Load Geometry from the File menu or use the Load Geometry button from the DeskProto button bar.

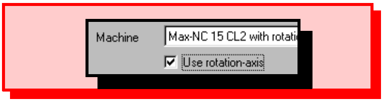
The geometry file that you have to load is called Venus.stl. Note that this file is not installed during setup. You will first have to copy the file from the DeskProto installation CD (section Bonus geometry) to your DeskProto data directory on hard disk (by default "C:\Program Files\DeskProto 5.0\Data", however you may of course use any directory). Trial version users (having no CD) can download the file at www.deskproto.com The file is only available in mm, so inch users will have to scale down with factor 0.03937

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 tailstock is available to support the far end. DeskProto only supports this type of rotation axis.
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 (vertical 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 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 (at least for most machines) as the model is full size (in mm).

In addition to **Rotation axis machining** as explained in this lesson, where the part rotates during machining, DeskProto also supports **Indexed machining**. Here the part is fixtured onto the rotation axis too, however plain XYZ toolpaths are applied, in a number of operations from different sides, with a positioning rotation in-between each two operations. Use the N–Sided Wizard for this aim.

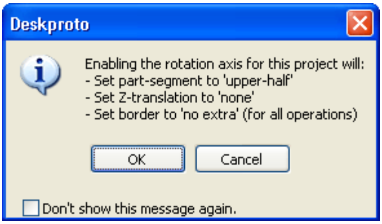
Using the Rotation axis

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 machine definition in DeskProto's Library of Machines it must have been configured.



Switch on the option Use Rotation-axis in the Part Parameters dialog. If this option is grayed out your machine definition does not have a rotation axis.

When you check the option Use Rotation axis the messagebox shown below will pop up. It tells you that a few parameters will be automatically changed to accommodate rotation axis machining. These parameters will be discussed later.



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: 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.

Probably you do not see this as DeskProto is complaining that the model does not fit inside the working area of the machine, which does make sense as the unscaled segment is a cylinder of 332 mm diameter. So you will have to **scale down**, the scale factor to be used depending on your machine's dimensions. Not only must the segment fit inside the working area, it also has to fit in the rotation axis unit. There will only be a limited distance between the rotation axis and the working table below, and the radius of the segment may not be larger than this distance. Otherwise of course the block defined by the segment cannot rotate. 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.

As the object is rotated around an axis at position (Y=0,Z=0), it might be that the rotation axis is in fact outside the geometry, resulting in a very large cylinder being drawn as the segment. This is in fact the case with the Venus example: the segment is not correct, and changing the segment values does not help. For these cases an extra option is present: 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.

A very important issue is the position of the **workpiece zero point**. For rotation axis machining two different conventions do exist for Z: the workpiece zero point either can be either on the rotation axis (so inside the block) or on the outside surface of the (cylinder) block. You can change the X-position of the zero point as usual, using the Translation option, however the Y cannot be changed and for Z fewer options are available. 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 messagebox just shown, DeskProto has already automatically set the Z Translation to None. The result can be checked in the Geometry Information dialog (Tab page Part) and/or by drawing an “Orientator on the workpiece zero point”. Make sure to set the workpiece zero point on your machine accordingly: an incorrect setting may damage part, cutter and/or machine !!

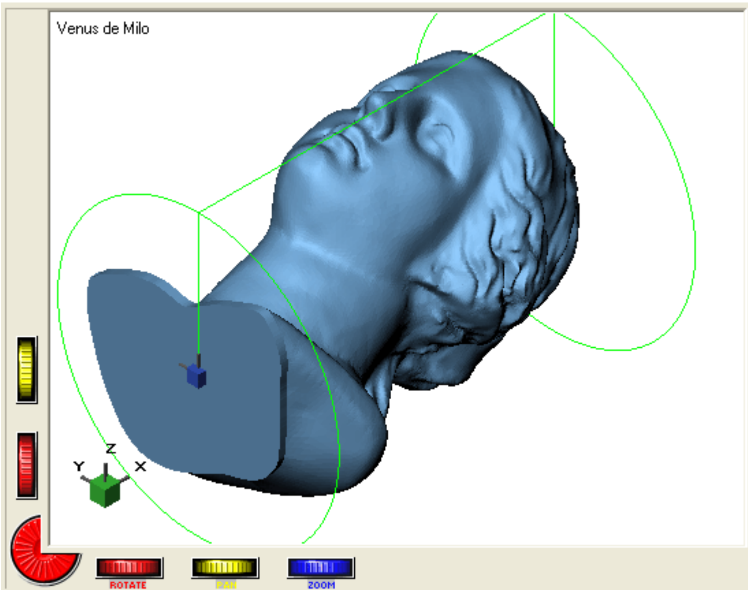
The other parameters that have been changed automatically are:
The Part segment has been set to Upper half only: as the geometry is rotated the bottom half will be on top afer 180 degrees, so upper half is sufficient to machine the complete model (for most geometries).
The Borders have been set to ‘No extra’, for all operations. You can find this parameter on the Borders tab of the Operation parameters. It’s 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 tailstock. So the outside surfaces left and right may not be machined, which is achieved by this setting.

When the rotation axis is used, for some parameters in DeskProto the Y-axis setting (to be set in mm) has been replaced by the A-axis (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 or XAZ, 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.

As a result of all this, you should now see the following picture on the screen:



In review, in order to get the result as shown above, 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
- Set the translation for Z to “none”

In the illustration do note the blue block at the bottom surface of the statue: this is the extra orientator that indicates the workpiece zero point. As you have set the Z translation to None this orientator is on the rotation axis.
In case you do not see the blue block: it can be switched on in the Subjects dialog as explained in Lesson 1.

Fine-tuning the parameters

Basically all 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-axis direction the Distance between the 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. 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 always some connection remains 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). You then can play with the Max X value of your segment in order to machine the complete right side (top of the Venus head) or not, depending on whether you want to use a tailstock or not. In fact by enlarging the segment you just manually add a border area at that one side only.

Without such flat surface you may want to define a connection shaft: a cylinder shape where the cutter may not come. Easiest is to define such connection shaft 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 while machining the model the cutter cannot go below this minimum Z-level, so for certain models some are will remain not machined.

The connection shaft can also be defined using one or two cylindrical support blocks. That is how the Rotation axis wizard works. Note that then the Borders need to be switched off in order to prevent machining the flat outside surfaces of the support blocks.

For this Venus lesson do switch off the Borders, and also make the Max X segment dimension a bit smaller to prevent the cutter damaging your tailstock in case you use one.

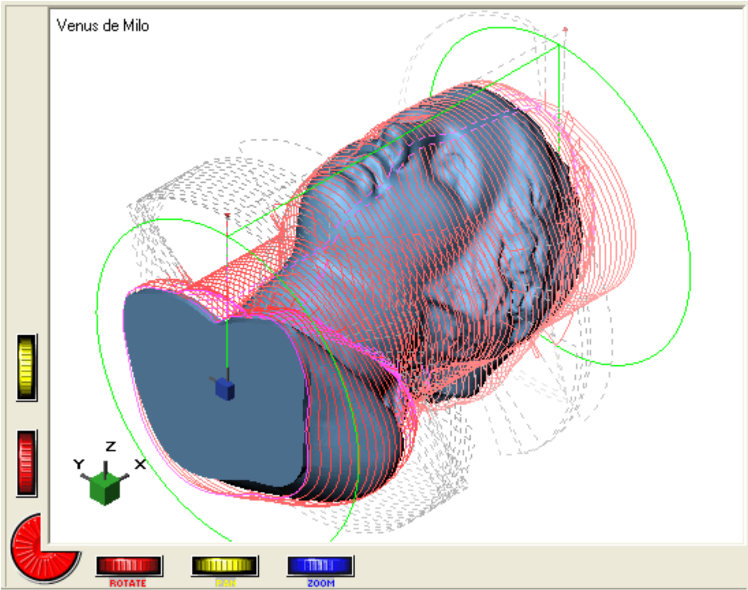
When rotation axis machining the Z-segment boundaries do require extra attention: both Z-min and Z-max. As said the Z-min can be used to define the connection shaft, unless part of the geometry is lower and would be skipped then. In fact part of the geometry can even be below Z=0, then you have to choose a Min Z value below Z=0. This is needed in case the geometry is ‘excentric’ the rotation axis line is partially outside the part. Then for certain angles the complete part is below the axis. Imagine a curved banana being machined: at a certain angle the middle part of the banana will be completely below the rotation axis line. 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 Max-Z segment boundary requires extra attention as well. In many cases you will prepare your block of material using a bandsaw machine: resulting in a rectangular block. The cylinder needed has to fit in this block, and so the block’s maximum diameter (the diagonal line of the block) will be much larger than in the DeskProto calculations. 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.

Finally: perhaps you did have the bright idea to use toolpaths in the Y (A)-direction, combined with either conventional or climb machining. This way the rotation axis can remain rotating in the same direction, 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 move up the tool to Zfree safe height and rewind the full 360 degrees after each revolution of the rotary axis.

When your machine does support continuous rotation in one direction, then you can select this in the machine definition: 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 one model but may prove to be useful later.



The resulting toolpaths will look like this, depending on your choices for scaling, cutter and precision. Do note that you can clearly recognize the first layer, which is a cylinder except where the geometry is higher. The bottom surface of the statue remains un-machined: the connection with the rotation axis. The toolpaths shown are in A direction and meandering, so the cutter does not need to rise to Z-free level there to rotate back 360 degrees.

To the milling machine

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

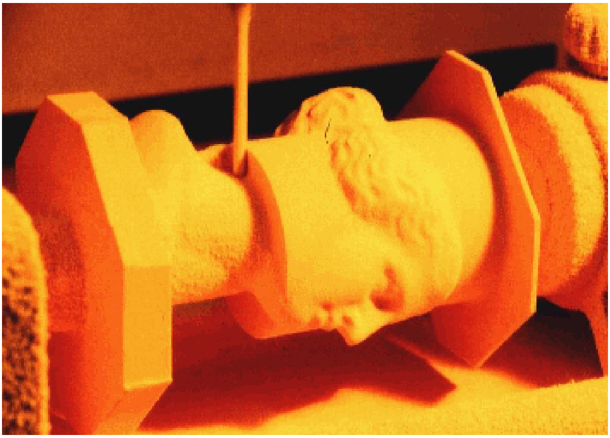
First you will have to fixture your material block, using the available options of the rotation axis, like a 3-jaw chuck, pins on a circular plate or a drill-head. Use the tailstock with a centering pin on the opposite side if one is available, as this makes the stability of the model much better. For 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.

Next you have to set the workpiece zero point. When you use translation method “Make top of part zero”, the procedure is the same as in the previous chapters: move the tip of the cutter to the desired workpiece zero point, and tell the machine that this has to be point (0,0,0). It now is critical that the cylinder block has exactly the correct diameter: otherwise the resulting part will be too thick. The Y=0 needs to be with the tip of the cutter exactly above the rotation axis, the Z=0 with the tip of the cutter touching the material.

This is more complex when you use translation method “None”, as the zero point is now inside the material block. On some machines the Y and Z of the rotation axis will be fixed and known: then these values can be used to accurately set zero.
If not you will have to manually set these: a good way is to machine a simple cylinder in your block using hand control, with the tool set above the rotation axis (Y-value), and on a known Z-value. Then measure the radius R of the resulting cylinder, and the point Z=0 will be R mm below the Z used for cutting. Y=0 then can be determined by letting the cutter touch both the front and the back of the cylinder and calculating the Y value of the halfway point.

Anyway with this zero position do 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.

Many controllers have problems in setting the correct Feedrate when rotation axis machining, as the rotation speed to be applied 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 might be (very) incorrect.



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 result will look like this. Do note the cylinder form created by the first layer. After machining you can remove both sides of the block using a simple bandsaw.

Lesson four
The Cellphone

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 DeskProto Lite.

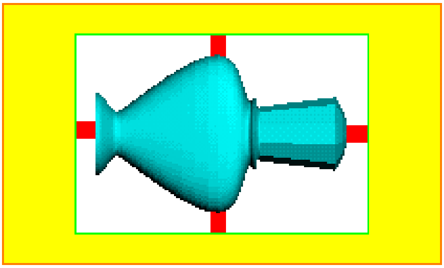
The example used is (again) a very fine geometry: the front panel of a cellphone. It is not from an existing phone, but has been specially modeled for this lesson by John Brock of Robert McNeel & Associates, using the Rhino 3D CAD software that McNeel manufactures. Also the rendered picture above is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

The cellphone STL is a bit large file (8 Mb), so it is not included in the standard DeskProto setup. You will have to get the file from the Bonus geometry section of the DeskProto CD or download it from www.deskproto.com , and then copy it to your DeskProto data directory.

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 most users.

The geometry file that you have to load is called 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) to your DeskProto data directory (by default "C:\Program Files\DeskProto 5.0\Data", however you may of course use any directory). Trial version users without CD can also 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).



Generally speaking, for milling from two sides the problem is the second side: how to fixture the model in the correct orientation to machine the second side, and how to re-establish the workpiece zero point for the second side. 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 is it easy to re-fixture the model for the second side, and the wizard also uses the frame to set a repeatable workpiece zero point.

Start a Project using the Wizard

In case the DeskProto wizard does not pop up automatically on start, you can call it from the File menu..

Start the DeskProto Wizard (File menu) in case it did not show up automatically after starting DeskProto. Choose **Two-Sided Milling** and press Next (the use of this Next button will not be mentioned in the rest of this lesson). **Load** the file Cellphone.stl. The wizard will next ask you if a geometry rotation is needed, assisting you by drawing the geometry in four standard views and by showing the geometry information dialog. For the Cellphone **rotating** is not needed.

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). Do note the light yellow banner line on top of each wizard page: this line indicates where the functionality can be found without the wizard.

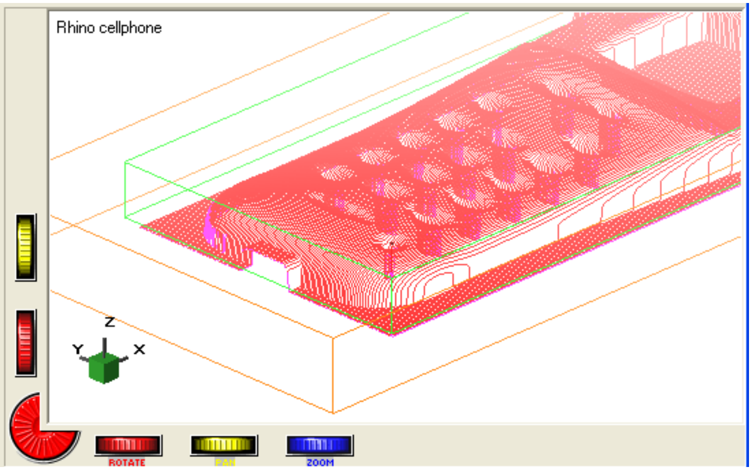
Which **cutter** to choose is an optimization between cutting speed (a larger cutter can remove material more quickly), surface quality (a larger ballnose cutter will create a smoother surface), small details (for small inner radii a small cutter is needed), and height of the model (small cutters are short). For this cellphone part we advise you to use a 3 mm ballnose cutter (radius 1.5). For inch users an 1/8 inch diameter ballnose will be a good choice. In general ballnose cutters are best for these freeform surfaces: when using a flat end-mill every toolpath will be clearly visible (as a “stair step”) at sloping areas. 3 mm diameter is sufficient for most details here, and the cutter is long enough to machine the complete model. Later you are free to add extra detailing operations with a smaller tool.

The **precision** to be used depends on your needs (and obviously also on how fast your machine is). The suggested value of 0.6 mm for both path distance and stepsize results in a rough model. In this lesson we will use 0.33 mm (or 0.0139"). You can also enter a high value here for roughing, and add a finishing operation later.

Next step is adding the **support blocks**, as explained above. The wizard will just add four default blocks, at the minimum and maximum X and Y values of the part. The thickness of these bridges (support blocks) depends on the size of your model, their length depends on the cutter that you have just chosen (the bridge must be long enough for the cutter to move around the model). So when you want to use more than one cutter start with the thickest one in the wizard. After finishing the wizard you can if needed change to Custom support blocks, to be positioned and sized exactly as needed (Part parameters, tab Support). The support blocks are added to the geometry: this adds an extra 12 triangles (two per side) for each support bridge.

The **size of the material block** is set by the wizard: both right and left 25 mm (1") is added, on front and back 10 mm (½") (You can change these values in the Preferences). Right and left more, as room is needed for the clamps to fixture the block on the machine. You can only choose the Z-dimension: 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 block will be drawn on screen as line drawing: 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.



The first part, for the first side, is ready now, and so you can **Calculate the toolpaths**. The illustration above shows part of the result. To make this illustration a bit clearer a long cutter has been used (on your screen two layers may be present as the cutter cannot cut sufficiently deep in one go). Also the geometry is not shown in the illustration. In the toolpath drawing you can clearly see that the outside faces of the support blocks are 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’ (so the machined cavity will be the same size for any cutter diameter). When the toolpaths look OK you can **Write the NC program file**. Do choose a filename that indicates which side of the part it is for.

The wizard continues with the second side now, for which a second part has to be created. The wizard will **Copy the part** to create the second part, as then all settings made for the first part will be copied to the second part. Only a 180 degrees geometry rotation is added, resulting in the part being turned upside down (rotated around the X-axis).

As a result the second part is immediately ready, and you can proceed to calculate toolpaths for the second side as well, and also write the second NC program file.

The final step of the wizard is to **Create a report**. 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 print this file and save it for future reference.

Fine-tuning 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”.

Earlier in this lesson it was already suggested to later add extra operations for finishing and detailing (as you started with the thickest cutter, the operation done by the wizard will be the roughing operation). After the two-sided wizard you have to be careful though: some changes will ruin you two-sided milling setup. What you must NOT change is the block size (so both Part segments) as this size is needed to exactly match both sides and set the workpiece zero point. Also the area to be machined (so X and Y of the operation sub-segments) can best be kept as set, it may anyway not be enlarged. The minimum Z of each operation sub-segment may be changed in order to minimize cutting time. The best way to add an operation is by **Copying** the existing operation in the Part Parameters dialog, as then the settings for sub-segment and Borders will be copied as well. All other operation parameters can be changed as needed.

The support blocks may be changed: you can only do so for the first part. Obviously the support blocks for both sides must exactly match, so to make sure that the same changes are made for both parts, in the second part DeskProto has selected the option “Use bridges of first part”.

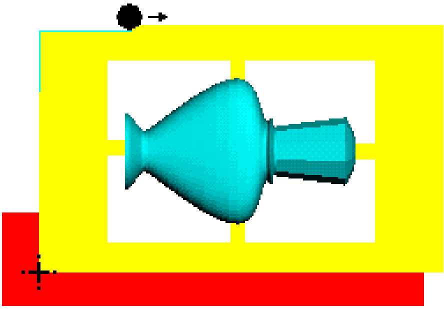
What we did for our own Cellphone show-model is (as said) create a roughing and a finishing operation for each side, using the same cutter. This way the finishing operation does not have to remove much material (the skin only) resulting in a high surface quality.

For the second side we also added an operation with a flat tool (2 mm diameter), as the inside geometry contains many sharp inner corners. Finally we added two detailing operations with a very small tool (diameter 0.8 mm or 0.03") for the microphone and speaker slits. Here we also changed the operation sub-segment to only machine a very small area. As the Part segment is the same for all operations, the workpiece zero point will be the same as well. The resulting 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 these for the first side.

For this aim DeskProto uses two rulers (stop-bars) that are exactly parallel to the machine’s axes, and have a known position. See the illustration below: the long ‘horizontal’ bar is parallel to X, at the known Y position “Y-bar”, the short ‘vertical’ bar has a known X-position “X-bar” (this one is short, so being parallel is less important here). An easy way to create these rulers is to just machine them using manual control: that way they are sure to be parallel, and their positions are exact. Also see the illustration below.



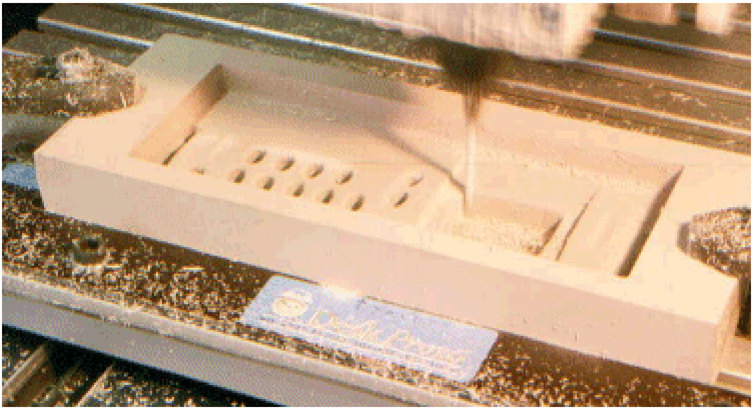
The workpiece zero point for both NC program files has to be set at (X-bar, Y-bar), with the Z=0 set with the tip of the tool touching the top of the block. As said for both files: the zero point remains the same after turning the block upside down for the second side.

After machining the first side, you have to machine two reference planes on the block's back and left side: see the cutter in the illustration above. The reference plane on the left is on position X =0, at the limit established by the X-bar, 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.

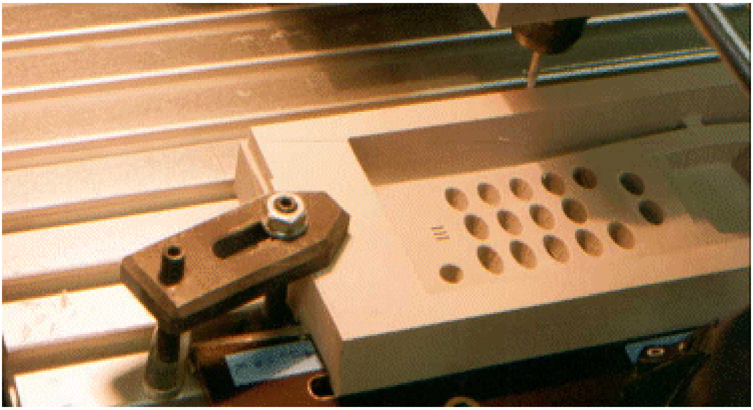
After turning the block upside-down these two reference planes will touch the rulers, making sure that the block is exactly lined up with the machine, and exactly at the correct position.

As an alternative to the **Ruler /Reference plane** method just described, more 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 a few more illustrations that show this process for the Cellphone model.



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

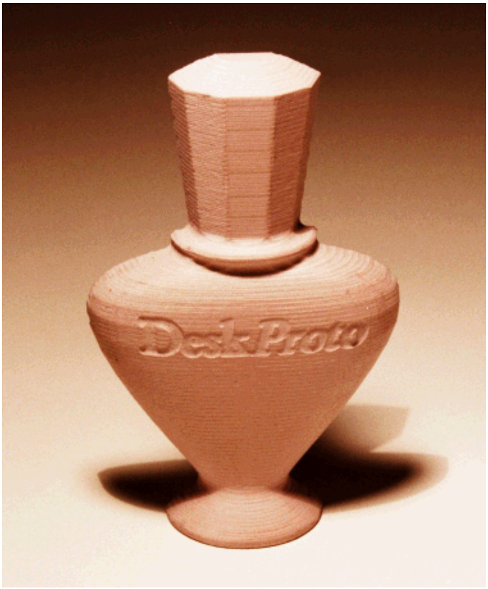


Before loosening the clamps, first you have to machine two reference planes on the left and on the back, as shown here. This is why some oversize is needed on the back side of the block. Also the left side of the block needs to be a bit inclined to have some excess material to be machined.

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

The resulting cellphone front cover, fresh from the machine without any extra work done. Do note the reference planes and the support blocks (bridges): both clearly visible. The frame and bridges have to be removed manually, and some sanding is needed where the support blocks were attached to the model.

Lesson five
2D Machining

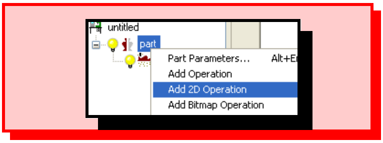


The lessons so far have all been addressing DeskProto's main application: creating 3D prototypes. As additional functionality DeskProto also offers some basic options for 2D machining. You have to be aware that DeskProto is not 2D CAM software: when you need options like 'pocketing' and cutter compensation 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 Milling Wizard** to teach you the basics of 2D machining.

Creating a 2D Operation

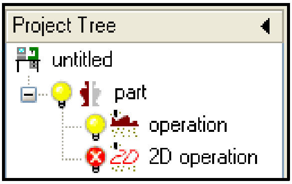
Start this lesson with creating a new project, using either the command New in the File menu, or the New-button on the menu-bar. Do not yet load any geometry file: though in most cases 2D machining will be used as extra functionality in a 3D project, it is also possible to use DeskProto for some simple 2D machining without 3D geometry.



Click the right mouse button on the second line of the project tree (the line showing 'part'), and then click 'Add 2D Operation'.

The alternative method for creating a 2D Operation is in the Part Parameters dialog. This dialog contains buttons to Add, Copy and Remove Operations. The Add button can be changed using the black arrow at it’s right: you can choose between Add 3D, Add 2D and Add bitmap for the type of Operation to be added.

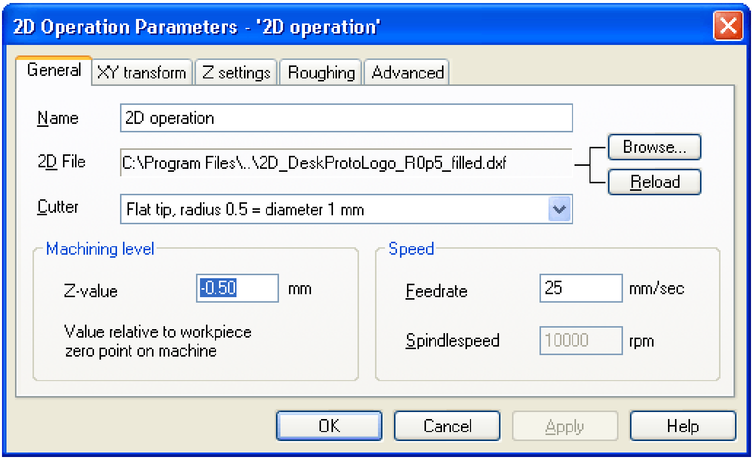
You will see that after adding the operation a new line appears in the tree, with a different icon, showing the new 2D Operation. This line starts in Edit mode, allowing you to give then new Operation a proper name. As shown in the illustration below, the lamp icon in the Tree is in **red** indicating 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.



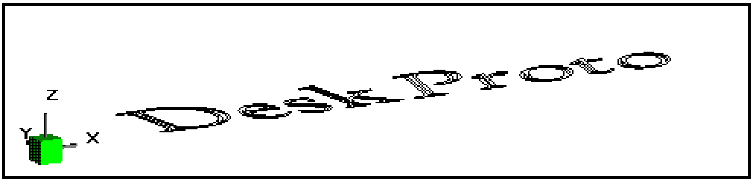
If you want to create only 2D toolpaths you now need to remove the 3D Operation (right mouse-click + Remove), as DeskProto will refuse to calculate toolpaths for a 3D operation if no 3D geometry has been loaded.

Opening a 2D file

Double-click on the new line in the tree to open the 2D Operation parameters dialog. Alternative is to use the command Operation parameters in the Parameters dialog. The dialog **2D Operation Parameters** will now open, which will look familiar as some 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. 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 filetypes 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.
All in 2D: any Z coordinate in the file will be ignored. Currently DeskProto will convert arcs to polylines. A point in the 2D file will result in drilling a hole on that location.

2D Operation parameter settings

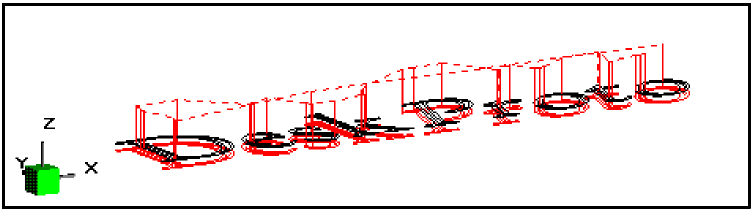
Now re-open the dialog “2D Operation Parameters” that was just closed, and look at the available parameter settings.

The parameters Name, Cutter, Feedrate and Spindlespeed are completely identical to their counterparts in a 3D Operation. For the Cutter there is one important difference though: **for 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.

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. At pen-down level the pen will draw a line, and in-between drawing moves the pen-up level will be used for positioning moves. The same happens while 2D machining: the Machining level Z-value defines pen-down, and the Free Movement height on the third tab defines pen-up.

Set the machining level to -0.5 mm (-0.02") and close the 2D Operation parameters dialog. Now calculate the toolpaths. The resulting toolpath can be easily interpreted.



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 the 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 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: for engraving such cutter will give you the best (sharpest) resulting characters.

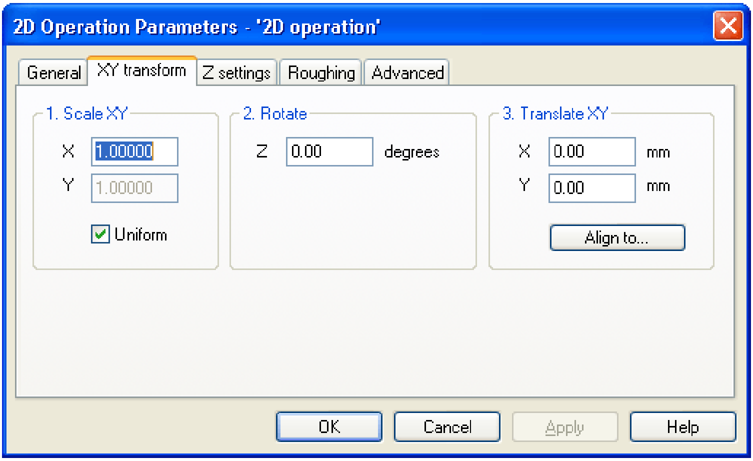
Combining 2D contours with 3D geometry

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 prototypes 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

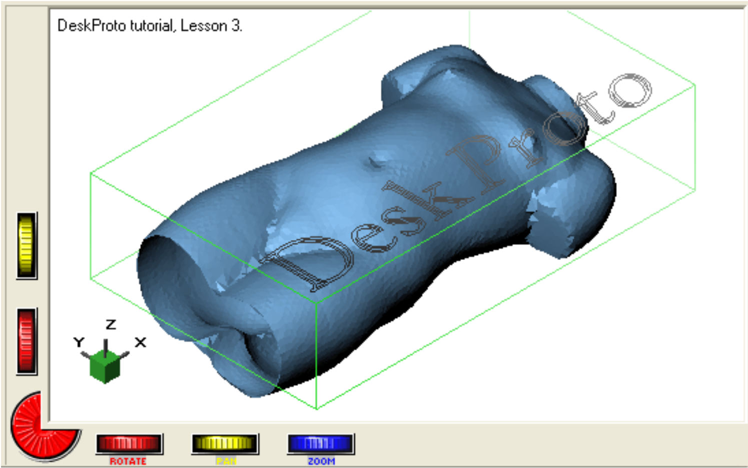
None for 3D Translation on all three axes. Your will then see that 2D and 3D files in DeskProto will be positioned just as in your CAD software.



In any other case you can position the 2D contour data using the 2D Operation parameters on Tab page “XY Transform” shown above. Their use will be clear: Scale to change the 2D size, Rotate to rotate 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 to see what you are doing.

Load project TORSO using the Open button from the DeskProto button bar. You need not save the project just created.

For this lesson we will use the example 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.
In Part Front add a 2D Operation, and in the 2D Operation parameters open the 2D File *2D_DeskProtoLogo_R0p5_filled.dxf*, as explained earlier in this lesson. You will see a picture as shown below.

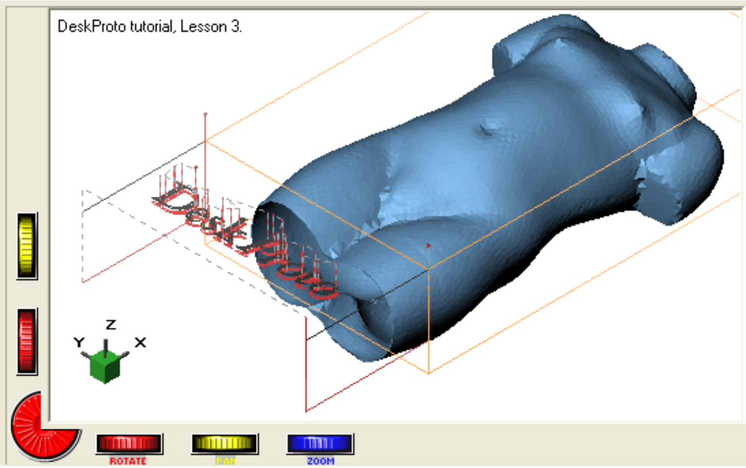


For a correct positioning of the logo on the Socle of the statue, the 2D contours will have to be rotated, translated and scaled. Open the 2D Operation parameters and use the Scale, Rotate and Translate options to position the 2D logo on the Socle to be created below the statue. The result should look like the picture on the next page

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.

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 there, 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 it 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; set a machining level of for instance minus 15 mm (-0.6"). The resulting screen will roughly look like the picture below.



In addition, 2D machining can also be used for drilling positioning holes in your part, for a correct positioning after turning upside-down (as an alternative to the ruler used in the DeskProto wizard).

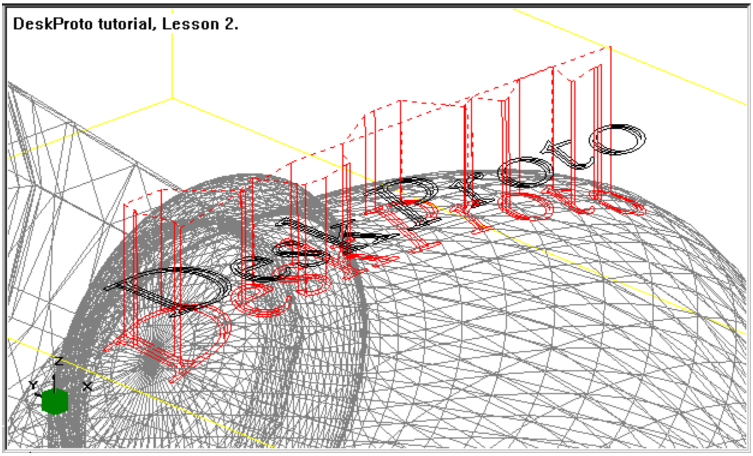
A nice detail is that 2D machining can also be used with rotation axis machining: the 2D drawing then will be wrapped around the 3D cylindrical segment, like a label around a jam-jar.

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 won’t be constant, but instead will be applied relative to the Z of the part-geometry. The result will indeed be projecting the 2D lines on the 3D part. Best explanation will be an example.

Open a new project
Load the geometry Bottle
Rotate the bottle minus 90 degrees around X
Set the segment to Upper half only.
Add a 2D Operation
Load the 2D File DeskProtoLogo.dxf
Scale the logo to fit the bottle
Position the 2D logo over the widest part of the bottle (2D Y-translation) and center it (2D X-translation).

Set the Machining level to minus 0.5 mm (minus 0.002 inch).
Check the option “Project 2D Contour on 3D Part geometry”
Calculate toolpaths.



As a result your DeskProto screen should look like this. Note that in the Subjects dialog the Geometry has been set to Wireframe instead of Rendered, as otherwise the 2D toolpaths would be obscured by the rendered geometry.

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 (gridsize 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 two extra remarks.
First: it can clearly be seen that a thicker cutter is 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: though not so clearly, it can also be seen that begin and end of the logo are engraved deeper than the middle part. This is due to the fact that DeskProto’s 2D toolpath calculations do not check for the cutter diameter. The pen-down machining level is calculated for the center of the cutter (the tooltip),

and so for non-horizontal surfaces it might be that more material is removed, that the actual groove is deeper than the machining level. Ultimately, for a vertical surface, the complete cutter radius will be removed, over the full height of the surface. So take care ! Also note that the projection used is vertical, which might distort the 2D logo: a circle projected on a tilted surface will become an oval.

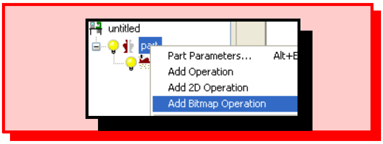
Lesson six
Bitmap Machining

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 3Dgeometry, and then calculates toolpaths over this geometry.

The bitmap used for this example shows the Austrian Fieldmarshal Radetsky (1766-1858). The bitmap file (greyscale 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

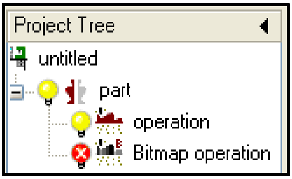
Start this lesson with creating a new project, using either the command New in the File menu, or the New-button on the menu-bar. Do not yet load any geometry file: as a first example we want to show milling only the bitmap, without 3D geometry.



Click the right mouse button on the second line of the project tree (the line showing 'part'), and then click 'Add Bitmap Operation'.

The alternative method for creating a Bitmap Operation is in the Part Parameters dialog. This dialog contains buttons to Add, Copy and Remove Operations. The Add button can be changed using the black arrow at it's right: you can choose between Add 3D, Add 2D and Add bitmap for the type of Operation to be added.

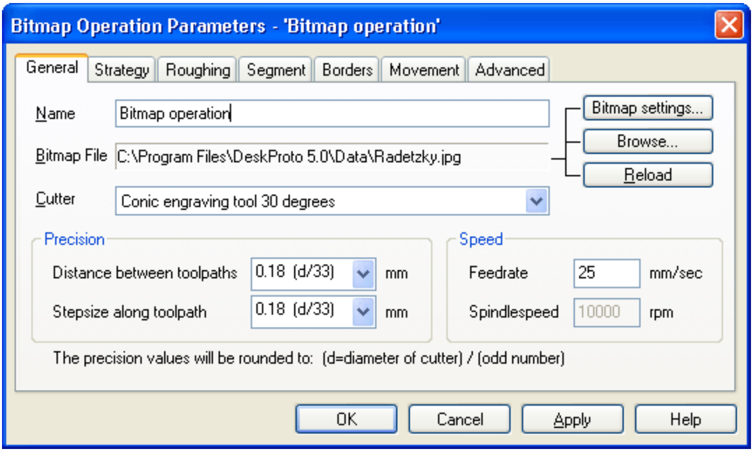
You will see that after adding the operation a new line appears in the tree, with a different icon, showing the new Bitmap Operation. This line starts in Edit mode, allowing you to give then new Operation a proper name. As shown in the illustration below, the lamp icon in the Tree is in **red** indicating that the operation is invalid. Reason is that you have not yet selected a bitmap file for this operation, and without such file a bitmap operation is invalid.



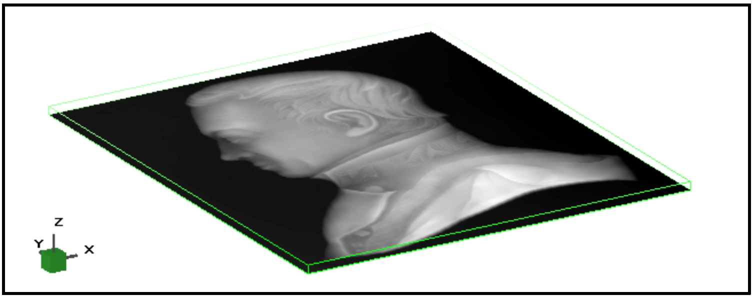
As you want to create only Bitmap toolpaths you now need to remove the 3D Operation (right mouse-click + Remove), as DeskProto will refuse to calculate toolpaths for a 3D operation if no 3D geometry has been loaded.

Opening a Bitmap file

Double-click on the new line in the tree to open the Bitmap Operation parameters dialog. Alternative is to use the command Operation parameters in the Parameters dialog. The dialog **Bitmap Operation Parameters** will now open, which will look familiar as most of the parameters are identical for Bitmap and for 3D.



In your dialog the entry for **Bitmap File** will be “none”. Use the Browse button at the right of the Bitmap File field 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: the result will look like this:



By rotating the view you can check that it is a pure 2D picture: one flat plane at constant Z-level, only the grey values vary. Note that the red lamp icon in the Tree has now become yellow (unless the default resulting relief is too large for your machine)

For Bitmap files DeskProto supports BMP, GIF and JPG files. Color pictures are automatically converted to grey values (Black and White pictures) as for the conversion to Z-levels grey values are needed.

Converting 2D bitmap data to a 3D relief

The conversion is in fact very simple: each pixel has a grey value, which can be black, white or some in-between shade of grey. This grey 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 a Grey-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.

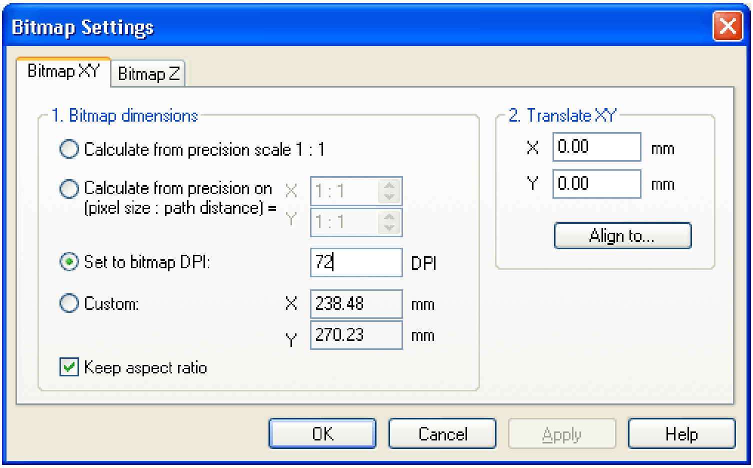
The Radetzky bitmap has been made using an existing relief and a 3D scanner. The shell bitmaps that you will find as examples also have been made using a 3D scanner, from real shells. When you can keep in mind how the grey-value to Z-height conversion is done, it is also possible to draw a picture in grey values for this application.

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. Two buttons have been added to this dialog: the Browse button that you have just used, 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 too 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 build the 3D geometry (the relief). For a ‘normal’ 3D Operation this is defined in the Part parameters, for the Bitmap operation it needs to be done on Operation level. For the relief you can set the dimensions (both XY and relief height) and the position. Press the button “Bitmap Settings...” to open this dialog.



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 options “Calculate from precision” need some explanation, especially as for those two options the quality of the resulting relief will be better than for the other two options.

The Precision in DeskProto is the Toolpath distance and the Stepsize along the toolpaths, as set in the (bitmap) Operation parameters. Calculation of the toolpaths is done using the Z-grid: a rectangular grid of XY positions, with a Z-value calculated for each position. The Precision sets the size of each grid-cell. Note that this algorithm resembles the Grey-value to Z-height conversion just described: both algorithms work with a XY grid with a Z-value for each position.

Combining two grids with different resolutions may lead to moiré patterns. In this case combining a Bitmap grid with a Z-grid of a different resolution may lead to ripples over the resulting relief, caused by the moiré effect.
You can understand why if you imagine a series of grid cells in de Z-grid: most of them containing the Z-value of one pixel in the bitmap, however every say tenth grid cell containing Z-values of two pixels as the size of once pixel is about 90% of the size of one grid-cell.

The size of the grid-cell is determined by the Precision, and the size of the pixel can be set here. So in order to prevent moiré problems it is best to select a pixel size dependant from the precision. This can be done using the two options “Calculate from precision”. The edit fields for Custom will show the resulting relief size.

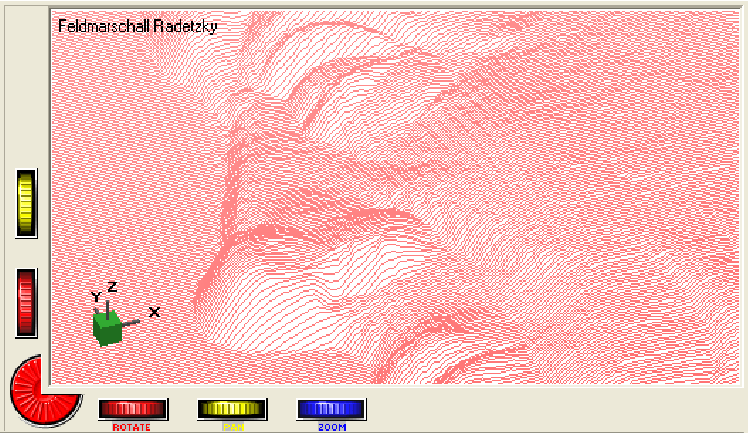
Select option “**Calculate from precision** on (pixel size : path distance) = “, and select a ratio that results in the relief size that you need. As you have set the Precision on 0.09 and want a relief size of about 50 mm (inch users about .004 and 2 inch respectively) you can choose **Scale 1:1**, which results in a relief size of 62 x 71 mm.

Note that when you later change the precision this will automatically also change the size of the resulting relief.

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 as these have been explained 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 will need to set a smaller value, like 0.2 inch.
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 below (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 (lesson 4) with a shell relief on the side of the bottle. Please do the following:

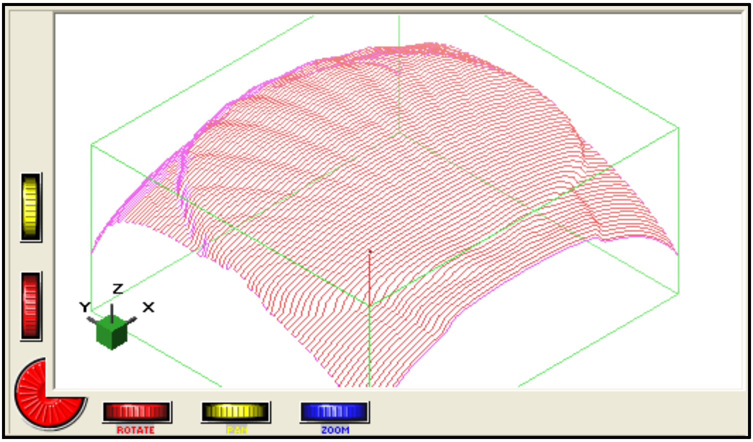
Start a New project
Load the geometry Bottle

Rotate the bottle -90 degrees round X
Set the segment to Upper half only
Add a Bitmap Operation
Browse the bitmap file *Shell1.jpg*
Select cutter “Ball nose, radius 1 = diameter 2 mm” (this will smoothen any 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 Calculate from precision on 2 : 1 (resulting size 33 x 30 mm)
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 use the Apply button to see what you do. We have selected a translation of 10 mm for X (bitmap in the centre) and 20 mm for Y.

Now select tab Bitmap Z.
For **Z calculation** choose 1.0 for White and 0.0 for Black. Note that the Translation in the (3D) Part Parameters must be on “Make top of part zero” for Z (the DeskProto default).
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 (though here for clarity less toolpaths have been drawn here).

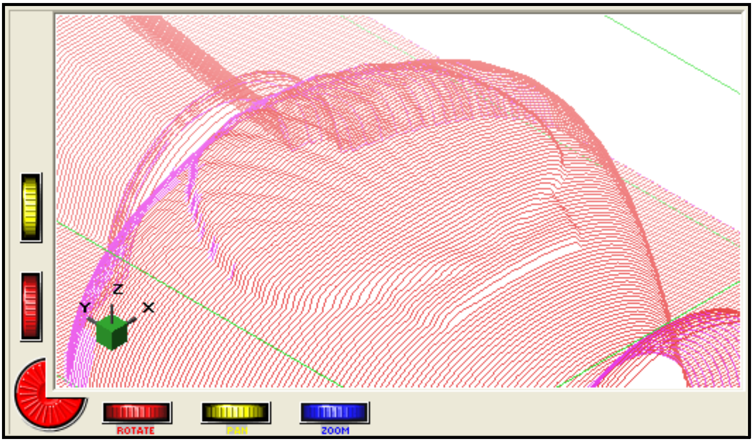


As you can see the bitmap relief has been projected onto the bottle geometry. This result would be sufficient for a relief to be engraved into the bottle, so for a relief with negative Z-values. However, in the example we have chosen positive Z-values, which leads to a serious complication: the first operation of this project (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.

This can be prevented by making all operations in this project bitmap operations: for the complete bottle (roughing and finishing) and for later detailing the relief with a thin cutter.

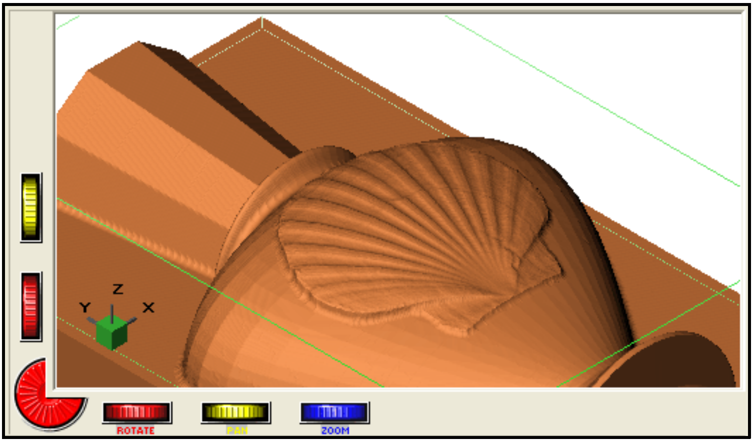
In order to make this Bitmap operation machine the complete part, two changes are needed in the bitmap operation parameters:
- For Borders select **Extra for Cutter**. In the picture above you can already see that the outside of the relief was not machined as for bitmaps the default for Borders is No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

After these two changes the toolpaths will look like in the picture below:



Again less toolpaths have been drawn for clarity of the printed picture.

You will probably want to use a thicker cutter and a larger distance between the toolpaths for the rest of the bottle model, especially when roughing. This is very well possible, you only have to take care as you have defined the relief size dependant on the precision (in order to prevent moiré patterns). To make sure that the relief is the same size for all operation you can best open the Bitmap settings and now choose Custom for the Bitmap Dimensions. The dimensions then will remain frozen on the size just selected, regardless of the precision of the operation.



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

Index

2D File . . . 71
2D Operation parameters . . . 71

A-axis . . . 47
Adhesive tape . . . 31

Bitmap . . . 79
Bonus geometry . . . 47, 59
Borders . . . 50, 53, 87

Calculate toolpath . . . 28, 41
Center geometry . . . 50
Clamp . . . 45, 65
Cutter . . . 13, 40

Data directory . . . 13
Distance between toolpaths . . . 41
Dongle . . . 7, 10
Drivers directory . . . 13
DXF file . . . 7

Facets . . . 19
Feedrate . . . 29, 41
Finishing . . . 42
Fixture the block . . . 31, 45, 55, 62

Geometry file . . . 18
Geometry Info . . . 26
Geometry parameters . . . 37
Geometry, rendered . . . 25
Geometry, wireframe . . . 25

Hardware . . . 7
High Chiploads . . . 44

Indexed machining . . . 48
Installation . . . 9

Layer . . . 42, 43

Load geometry 34

Machining level 72
Milling machine 31
Model dimensions 26
Mouse-wheel 22

NC program 30

Open existing project 18
Open new project 34
OpenGL 7

Pan 22
Part 15, 37
Part with more operations 42
Polygon data 19
Project 15, 36
Project 2D Contour 76

Rapid prototyping 7
Reference Pins 65
Reference plane 65
Right mouse button 16, 37
Rotating the geometry 38
Rotating the view 20, 39
Rotation axis 49
Roughing 42
Segment 39, 50, 62
Setup 9, 11
Skin 43
Step size 41
STL file 7
Strategy 44
Support blocks (bridges) 61

Troubleshooting 11
Two Sided Milling Wizard 60

Workpiece zero point 26, 31, 50, 60

Zoom 21