Desk*Proto*

Tutorial

Including Installation, Quick Start and **nine** Lessons.

Desktop Prototyping software,
to quickly create parts using a (desktop) CNC milling machine.

Version 7.0
Revision covering Windows, MacOS and Linux

Delft Spline Systems
Utrecht, The Netherlands.
Internet www.deskproto.com

Table of Contents

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 toolpaths before sending them to the machine - in case you are a novice user have an experienced colleague check them.

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

Licenses and Copyrights

DeskProto uses the following external libraries (installed during Setup as DLL files):

The **Boost C++** libraries.
Copyright © 1998-2005, Beman Dawes, David Abrahams,
Copyright © 2004-2007, Rene Rivera.
Used and distributed under the Boost Software License V1.0.
www.boost.org

The **Crypto++** library of cryptographic algorithms
Copyright © 1995-2013, Wei Dai
Used and distributed under the Boost Software License V1.0.
www.cryptopp.com

The **HIDAPI** library for communication with HID devices.
Copyright © 2009, Alan Ott, Signal 11 Software,
used and distributed under the HIDAPI license.
github.com/signal11/hidapi

The **Minizip** library for reading and writing ZIP archives.
Copyright © 2017, Nathan Moinvaziri
used and distributed under the Minizip license.
github.com/nmoinvaz/minizip

The **QT** cross-platform application framework.
Copyright © 2016, The QT Company Ltd. and other contributors
used and distributed under the GNU Lesser General Public License (LGPLv3).
www.qt.io

The complete license texts for all these libraries can be found in the DeskProto About box.

Essentials

What does DeskProto offer

DeskProto is a CAM program (**C**omputer **A**ided **M**anufacturing) for 3-axis, 4-axis and 5-axis CNC milling machines, offering **Desk**top **Proto**typing. DeskProto will allow you to machine 2D vector drawings, 3D geometries, as well as 3D reliefs based on photos. It can be used for product design, jewelry, woodworking, medical applications, arts, education, hobby, etc. DeskProto can be combined with any CAD program, and with any CNC milling machine.

Four different editions of DeskProto are available: **Free**, **Entry**, **Expert** and **Multi-Axis**, offering different (sub)sets of DeskProto's functionality. An edition comparison table can be found on www.deskproto.com

How does it work

The starting point for DeskProto is a CAD file (it is not possible to design in DeskProto: CAM is about calculating toolpaths). Three types of CAD data are supported, each with a slightly different work-flow:
Vector-data: 2D drawing containing lines and arcs, stored as DXF, AI or EPS file.
Geometry-data: 3D geometry defined as a collection of triangles (facets) that describe its outer surface (polygon data), stored as STL or DXF file.
Bitmap-data: 2D image containing colored pixels, stored as BMP, JPG, GIF, PNG or TIF file.
So in fact DeskProto offers **three CAM programs for the price of one !**

DeskProto will load the CAD file and display its contents. It is possible to load more than one file. At this point you can scale, translate, rotate etc. After entering some milling parameters (cutting tool, required accuracy, etc) DeskProto will calculate the toolpaths and save them in an NC file.
Send this NC file to your CNC milling machine and you will have your part *ready within a short time.*

What hardware/software is needed

DeskProto is available for Microsoft Windows, for Apple MacOS and for Linux.

For **Windows** it needs Win XP (SP3), Win Vista, Win7, Win8, Win10 or newer.

On 64 bits Windows versions a 64 bits DeskProto will be installed, otherwise a 32 bits version. Minimum required hardware is a Pentium PC with 1 GB RAM and 100 GB free disk space: faster/more is better. The graphics card needs to support OpenGL V2.1 or newer.

For **MacOS** (Apple) it needs Sierra (10.12) High Sierra (10.13), Mojave (10.14), Catalina (10.15) or newer. DeskProto for MacOS always is 64 bits. Here as well minimum hardware requirement is 1 GB RAM and 100 GB free disk space: faster/more is better.

For **Linux** it has been developed and tested using Ubuntu 18.04 (64 bits), still it should also work on most other popular Linux distributions. Only the 64 bits versions are supported. Here as well minimum hardware requirement is 1 GB RAM and 100 GB free disk space: faster/more is better.

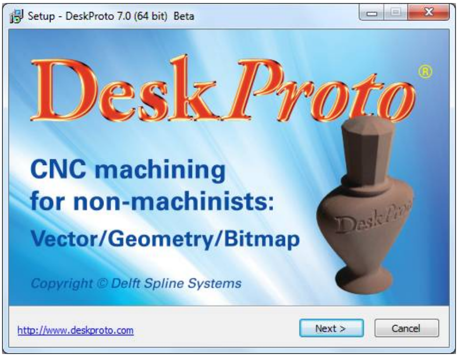
The screenshots in this Tutorial have been made using Windows, still for MacOS users and for Linux users these images will be completely clear as all screens are very similar when using one of these operating systems.

Installation

Windows: Setup file

DeskProto Version 7 runs in MS Windows Vista, Win7, 8, 10 or newer. Email us to obtain a Setup for Win XP (SP3). Minimum required hardware is a Pentium PC with 1 GB RAM: faster/more is better. The use of an OpenGL (V2.1 or better) compatible 3D graphics card is required. For installation you need about 100 Mb of free disk space, to use DeskProto you need much more for the NC program files that you will create.

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



Now just follow the instructions given:

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

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

When downloading DeskProto from the DeskProto website some browsers will start installing immediately, while other browsers will first save the Setup file on your hard disk. The DeskProto Setup program is called “dp70-en.exe” (in which “en” can also show a different language, like fr, it, ja, ...). It can be started just as any other Windows program.

When you have ordered DeskProto on CD you can simply insert the CD into your PC: a program called the DeskProto Install Menu will automatically start. In case that does not happen you can locate the file Setup.exe on the CD and manually start it. Note that the Setup program on CD has a different name.

MacOS: DMG file

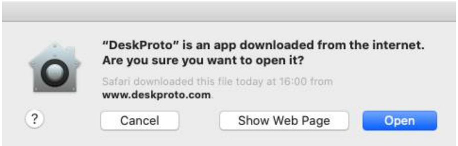
DeskProto Version 7 runs in MacOS Sierra, Mojave, Catalina or newer (for Windows users: MacOS is the software that runs on Apple computers). DeskProto for MacOS always is 64 bits. Minimum hardware requirement is 1 GB RAM and 100 GB free disk space: faster/more is better.

The file to download is a DMG-file. This is an Apple **D**isk I**m**a**g**e file: a read-only file that can be mounted as a disk. You can see it as a ‘virtual CD’. It is a standard way to distribute MacOS apps. The file that you download is called **dp70-en.dmg** (in which “en” can also show a different language, like fr, it, ja, ...).

You can mount the DMG by double-clicking it. The dialog that then will pop up shows the license agreement for DeskProto. When you agree with the terms of this license you can click on button Agree and proceed. Now the DMG file will be verified and opened. It will show the DeskProto70Installer dialog:

Installing DeskProto can be done by simply dragging the DeskProto icon into your Applications folder. That is all: you can now start DeskProto from

Launchpad, like you would any other app.
The first time that you start DeskProto a dialog will pop up that asks if you are sure, as the app has been downloaded from the Internet:



It confirms that the file has been downloaded from www.deskproto.com, and that a valid Apple Developer ID certificate has been found (if not a warning message would have been shown).

Linux: AppImage file

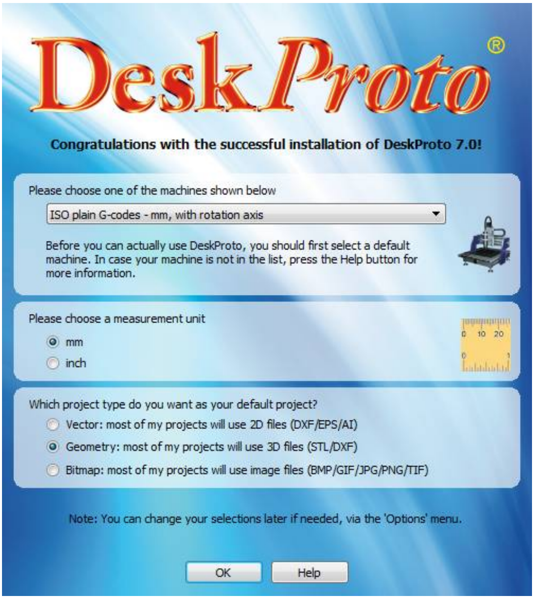
DeskProto Version 7 has been developed and tested using Ubuntu 18.04 (64 bits), still is should also work on most other popular Linux distributions. Only the 64 bits versions are supported. Here as well minimum hardware requirement is 1 GB RAM and 100 GB free disk space: faster/more is better.

The file to download is an AppImage file. This format is widely used for distributing Linux software as it is very easy to use: simply double-click this file to start the application. What happens (invisible for the user) is that the AppImage will be unpacked to a temporary location, from where the application is started. Similar to mounting a disk. The file that you download is called **DeskProto-7.0-en.AppImage** (in which "en" can also show a different language, like fr, it, ja, ...).

Important is that starting the application is possible only after you have changed the file's properties (right-click the file and select "Properties" to open a dialog like shown above). On tab Permissions of this dialog you need to check the option to Allow executing the file as a program.

Running DeskProto

The first time that you start DeskProto it will complete the installation by asking you which CNC milling machine you will be using, which units and which type of projects.

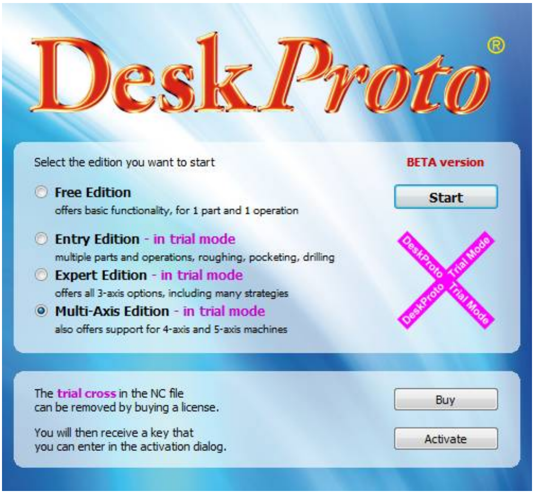


Select the correct **machine** (the one you have) in the drop-down list: it will be the default machine that DeskProto will use for all your projects. If needed this setting can later be changed in the default Project parameters (Options menu).

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

The **units** that you select here will be used for the CAD-data file import and for the user-interface. These are the units as set in the Preferences.
Units can be set on a second location as well: the units in the NC files that DeskProto writes are set in the postprocessor for that machine (the postprocessor is the software module that translates the output to the format needed by your machine, so a sort of Driver). Check the Help file for more information about the Units settings in DeskProto.

For each type of CAD-data (Vector, Geometry and Bitmap) different settings are required, so DeskProto features vector operations, geometry operations and bitmap operations. The default **project type** defines which operation will be loaded when creating a new project. As you can add and delete operations any time this is not a critical decision. If needed this setting can later be changed in the default Part parameters (Options menu).



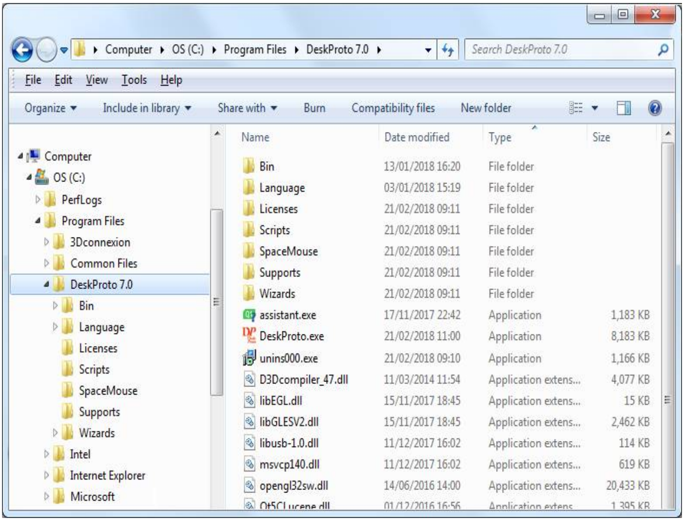
After this Setup you will have DeskProto running on your PC. You can immediately start using the **Free Edition**, which offers basic CAM functionality without having to buy a license. In the Edition select dialog (shown above) you can choose which edition you want to start.

More functionality is available in the other three editions, for which a paid license is required. On the DeskProto website you can find a Comparison table that shows the exact capabilities of each Edition. For users without a license these higher editions are available in **Trial mode**: both on the screen and in the NC toolpaths the **Trial cross watermark** will be visible.

When you have bought a DeskProto **license** you will have an unlocking code that you can use to remove this trial cross. In the dialog Activate License (Windows and Linux: Help menu, MacOS: DeskProto menu) you need to enter the Name and the Key on the license. Exactly as specified: with capitals, spaces and comma.

Files and Folders (Windows)

In the past DeskProto used to store all files clearly arranged within one directory structure. Unfortunately in newer versions of Windows that is no longer possible, due to the severe security rules by the User Account Control (UAC). Drivers and user data no longer may be stored on the same file-location as the program file.



By default DeskProto V7.0 stores its files in the these directories (folders):

\Program Files

\Program Files\DeskProto 7.0
This folder contains the file of DeskProto.exe and a large number of DLL files (program components) and other files that are used by DeskProto.
\Program Files\DeskProto 7.0\Bin
More program components that DeskProto needs
\Program Files\DeskProto 7.0\Language
This folder is used for all elements in the user interface that can be translated (resources, help, license, readme), and thus contains a sub-folder for each language. Sub-folder EN (English) is always present, other languages only in case installed.
Each language needs at least one resource file in this folder, after which in the DeskProto Preferences the new language can be selected. For instance for German a folder DE is needed with resource file file *deskproto_de.qm* (a qm

file is a Qt resource file). All other files in this folder are optional as for these DeskProto will default to English in case not found.
\Program Files\DeskProto 7.0\Licenses
This folder contains the licenses for all external libraries that DeskProto uses.
\Program Files\DeskProto 7.0\Scripts
This folder contains all Scripts, see the Reference manual (on your CD and on the DeskProto website) for more information.
\Program Files\DeskProto 7.0\SpaceMouse
Files needed for the 3Dconnexion SpaceMouse.
\Program Files\DeskProto 7.0\Supports
This folder contains the Geometry support tabs, see the Reference manual (on your CD and on the DeskProto website) for more information.
\Program Files\DeskProto 7.0\Wizards
This folder contains all files for the Custom Wizards, see the Reference manual (on your CD and on the DeskProto website) for more information.

\ProgramData

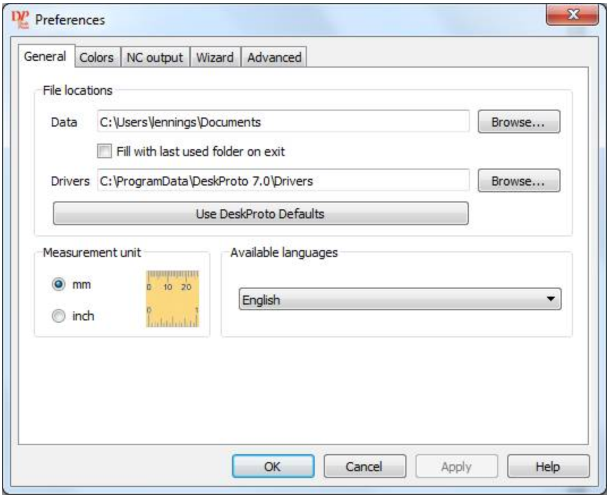
\ProgramData\DeskProto 7.0\Drivers
For making an NC program that is suitable for your milling machine, DeskProto needs information about the correct machine, the postprocessor and the available cutters. This information is available from the configuration files (*.MCH for the machines, *.PPR for the postprocessors and *.CTR for the cutters), stored in this drivers folder.
\ProgramData\DeskProto 7.0\Samples
For novice users each DeskProto comes with a number of sample files, also used for the lessons in this Tutorial. This concerns DeskProto Project files (*.DPJ), 3D Geometry files (*.STL), 2D and 3D Vector files (*.DXF), and a few sample bitmap files.

The location of the folder **\ProgramData** is different per Windows version.
In Windows XP this folder is called:
C:\Documents and Settings\All Users\Application Data\
For some reason this had been made a hidden folder: to make it visible in File Explorer ('My Computer”) open Tools >> Folder Options >> tab View and select the option “Show hidden files and directories”).
In Windows Vista and Win7, Win8 and Win10 the folder is located in the root (so as C:\ProgramData\ ). To make the folder visible:
Organize >> Folder Options >> tab View (Win Vista)
Tools >> Folder Options >> tab View (Win7)
use the View ribbon of File Explorer (Win8, Win10).

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

It is advised to create a folder structure here, for instance with a new folder for each new client or for each new project.

Some of the file locations are user-configurable:



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

Following Microsoft's guidelines DeskProto stores the Preferences in the **Registry**.

Files and Folders (MacOS)

The file DeskProto that you have installed is in fact a folder structure, containing both the program and a number of folders (for drivers, samples and much more). As application files are read-only it is needed to copy a number of these folders to a different location, DeskProto will do this automatically for you.

DeskProto V7.0 stores its files in the these folders:

~/Library/Application Support/Delft Spline Systems/DeskProto/7.0/
Here you can find the folders Drivers, Languages, Licenses, Samples, Scripts and Supports, the purpose of each folder has been explained on the previous pages. A folder Wizards is not present as custom wizards are not supported in

the MacOS version.

The DeskProto Preferences are stored in file
~/Library/Preferences/com.delft-spline-systems.DeskProto.plist

And the default folder for all user Data is
~/Documents/

The file locations for Data and for Drivers can be set in the DeskProto Preferences (DeskProto menu), see the screenshot in the paragraph on Windows. The button “Use DeskProto Defaults” can be used to reset these locations.

In the above folder specifications “~/” stands for your home folder.
This is in fact folder /Users/username/ - for instance /Users/john/ for user john. As a result changes that you make in these folders (like adding a new cutter) can be seen only by the current user. The license activation as well (stored in the Preferences) will only work for the current user. This is not a problem, still it is good to know that here the MacOS version differs from the Windows version.

Files and Folders (Linux)

As the AppImage file that you used to start DeskProto is read-only it is needed to copy a number of files and folders to a different location, in order to store your settings, Preferences, cutter definitions, etc. DeskProto will do this automatically for you.

DeskProto V7.0 stores its files in the these folders:

~/.local/share/Delft Spline Systems/DeskProto/7.0/
Here you can find the folders Drivers, Languages, Licenses, Samples, Scripts and Supports, the purpose of each folder has been explained on the previous pages. A folder Wizards is not present as custom wizards are not supported in the Linux version.

The DeskProto Preferences are stored in file
~/.config/Delft Spline Systems/DeskProto.conf

And the default folder for all user Data is
~/Documents/

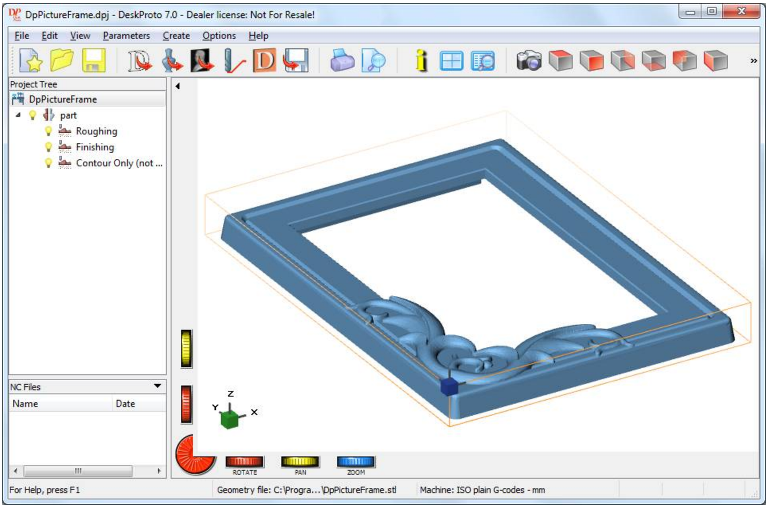
The file locations for Data and for Drivers can be set in the DeskProto Preferences (DeskProto menu), see the screenshot in the paragraph on Windows. The button “Use DeskProto Defaults” can be used to reset these locations.

In the above folder specifications “~/” stands for your home folder.
This is in fact folder /home/username/ - for instance /home/john/ for user john. As a result changes that you make in these folders (like adding a new cutter) can be seen only by the current user. The license activation as well (stored in the Preferences) will only work for the current user. This is not a problem, still it is good to know that here the Linux version differs from the Windows version.

Quick Start

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

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

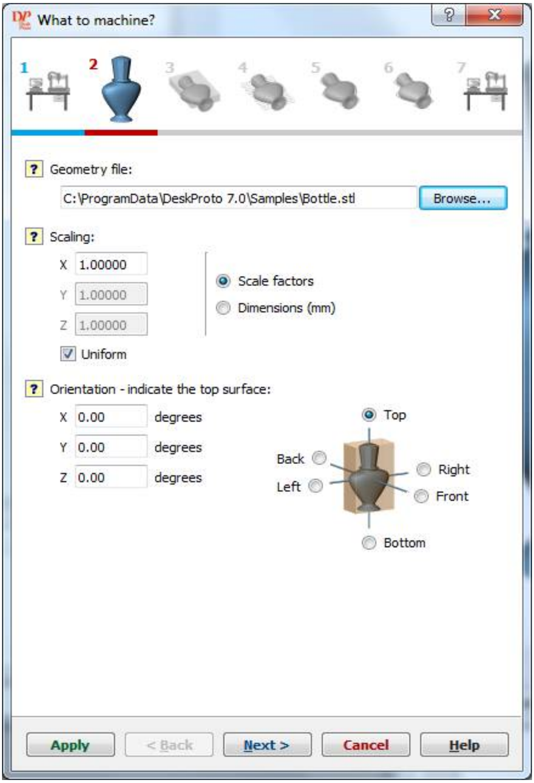


The DeskProto screen contains standard items like the title bar (top line), menu bar, toolbar (the row of buttons below the menu) and status bar (bottom line). The center area is divided into three tiles: the large View window on the right, and the windows Project Tree and NC files on the left. All these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

The above screenshot shows the **Windows** version of DeskProto. The **MacOS** version has an extra menu choice: the DeskProto menu, on the left side of File. It is visible on the top of the screen, so not in the DeskProto window. This is conform the MacOS standard and won’t confuse any Apple user.

The DeskProto menu in **Linux** is identical to the menu in Windows, and in most cases part of the DeskProto window. Though some Linux versions allow the menu to be shown at the top of the screen.

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



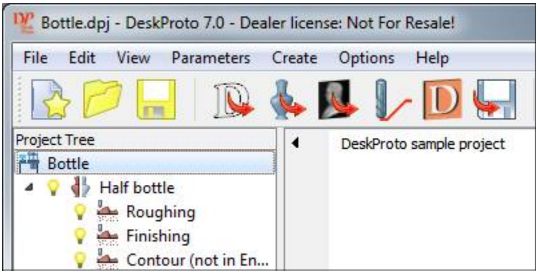
New users are advised to use the **DeskProto Wizards**, that will guide them through all the steps needed to generate an NC toolpath file using their own CAD-data. The illustration above shows a typical wizard page.

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

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

1. Project parameters include the name of the CAD-data file(s), the machine and the number of Parts that you want to use in this project.
2. Part parameters define **What will be machined.** They set the size, orientation, position and alike. Within each Part you can use one or more milling Operations.
3. Operation parameters define **How it will be machined.**
These are in fact the only real 'milling parameters'. Three different types of operation are available: Vector Operation, Geometry Operation and Bitmap Operation, as for these three types of data different settings are needed.

The **Project** is the central concept of DeskProto. All information about a project 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 references to the **CAD-files** (so the CAD-data is not included).



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

Note 1: four different editions of DeskProto are available: **Free**, **Entry**, **Expert** and **Multi-axis**. The editions Free, Entry and Expert contain subsets of the available Part and Operation parameters. Free and Entry do not offer Custom wizards and Scripts, and the Free edition is limited to using one part and one operation in a project.

Note 2: to open a CAD-file in DeskProto you have to use "**Load Vector file...**" "**Load Geometry file...**" or "**Load Bitmap file…**" in the File menu (if needed start a New project first). You cannot use File>Open, as you do not yet have a DeskProto project file for this new project. The CAD-data that you load will be available for all Parts and for all Operations.

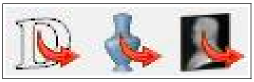
All function in DeskProto can be reached using the pull-down menus or using the buttons on the toolbar. The most important menus are described below:

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

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

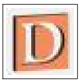
* The **Create Menu** is the most important; this is where you can start the milling calculations and write the NC program file.

The most important buttons for the DeskProto process are:



The first step is to **Load CAD data**, Vector, Geometry or Bitmap (or a combination).

After setting some parameters you can then **Calculate the toolpaths** and view them on the screen.



The **Simulation** offers a preview of what the part will look like.

Finally you **Write the NC file** and send it to your CNC milling machine.

We do hope you will enjoy using this software!

1. Beer Tray (basic vector)

Lesson One

In this first lesson the most elementary functions of DeskProto will be explained: you will learn about the DeskProto user-interface and its main functions. You will create toolpaths based on a 2D drawing and create an NC file, ready to be sent to the milling machine. The lesson will be presented **twice**: first using the **Wizard 'Basic Vector machining'** and next using the **Dialog-based interface.**
This lesson is for all DeskProto editions (though in the Free edition the text on the tray will be skipped).

The CAD-data for this project is a 2D drawing, so a drawing that does not contain any Z-values. The drawing contains only lines and arcs (**vector information**), saved in a DXF file. It contains the design of a Beer Tray: one outer contour with seven round holes and (optional) a text to be engraved. The tray may of course also be used for coffee :-).

Feel free to follow this lesson using your own drawing, and/or to replace the text in the sample DXF file by your own text.

Start DeskProto

You can start DeskProto most easily using its Program Icon on your desktop. When starting, DeskProto will briefly show its Splash screen. The first time you start DeskProto an extra dialog will pop up asking you to select the machine and the units to be used, see the Installation paragraph. After that DeskProto will show either the Edition select dialog or the Start screen.

The **Edition select** dialog (shown above, in chapter Installation) is shown only when you are running the DeskProto Free edition. It asks whether you want to start the Free edition or want to trial one of the other editions. In addition buttons are present to either Buy a license or to Activate one.

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

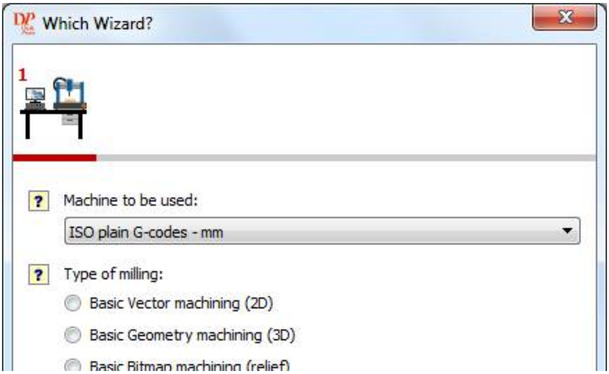
DeskProto offers two different user-interfaces:
Wizard-based and **Dialog-based**.
For novice users the Wizard interface is most important, as the Wizards will step-by-step guide you along all actions needed to create an NC file in DeskProto. All settings offered by the Wizards are also possible in the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

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

Lesson 1A

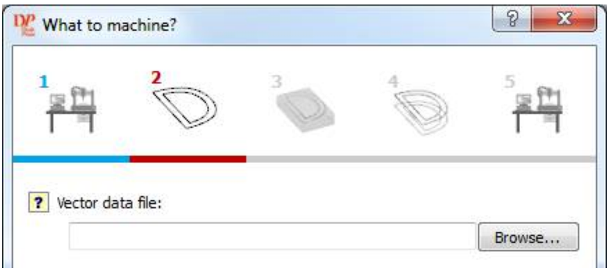
The Beer tray, wizard interface

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



The Machine to be used should already be the correct machine, as you have set your default machine when first starting DeskProto. If not correct you can select a different machine here (in order to change the default machine you need to open the Default Project parameters, via the Options menu).

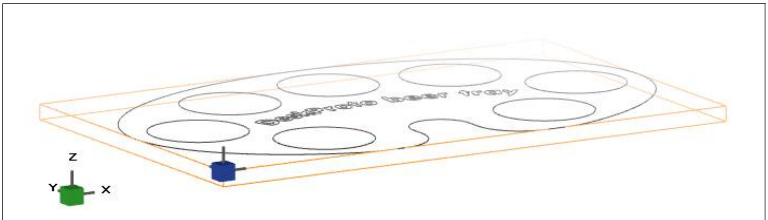
A series of different wizards is available, each meant for a specific type of milling. For the Beer tray we will use the first wizard: **Basic Vector machining**, that is available in all Editions of DeskProto. So please select that wizard and press Next.



As you can see above now the second icon in the dialog becomes active (enlarged and underlined in red): you are on the second page of this wizard. These icons can be used as navigation tabs: you can click on any of the colored icons (except page 1) to directly go to that wizard page. Most forward jumps are not possible though (gray icons), as you cannot skip any of the wizard's pages.

On this second page you first need to browse a **Vector data file**. As you started with option "Use samples folder" checked, the Browse button should directly look in the DeskProto Samples folder: select file *2D_DpBeerTray.dxf* and press Open.
Note for INCH users: *The file just mentioned is in mm, for a material slab of 400x200 mm. When loaded in your inch DeskProto this will be a slab of 400x200 inches: way too large. When you work in inches you need to select the inch version of this file, called* ***2D_DpBeerTray_inch.dxf****.*

In case you do not see the Samples folder: you can find it in folder \ProgramData\DeskProto 7.0\Samples\ (for more information see the earlier paragraph on Files and Folders).

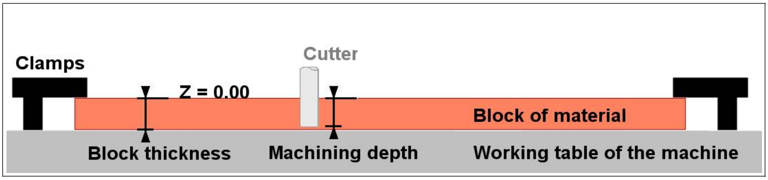


After opening the 2D_DpBeerTray file it will be drawn on your screen, see the image above. All vector information in gray, added in orange the material block, and two small cubes with axes which are called **Orientators**.
This default material block is the exact bounding box of the vector-data, with a thickness of 10 mm (or 0.5"). The green orientator shows the three axes: as you can see the largest size of this design is along X. The blue orientator shows the position of the workpiece zero point, more about that later.

In this lesson we will only deal the outer contour and the holes: this wizard only offers "basic vector machining". Engraving the text is a more advanced step that will be explained in Lesson 1B.

This wizard page offers options to Scale and Rotate this 2D drawing. **Scaling** to a smaller size may be needed when your machine is too small for this part. Unfortunately the result probably then will also be too small to carry beer glasses or coffee cups, so in that case you can better create your own drawing. **Rotating** can be useful if on your machine the Y-axis is the longest: then rotating 90 degrees may make the tray fit inside the machining area. As this is a 2D drawing you can only rotate round the Z-axis.

For more information on any of the settings that the wizard offers: position your cursor over one of the **yellow question marks** to get an explanation: a Wizard tooltip then will pop up.



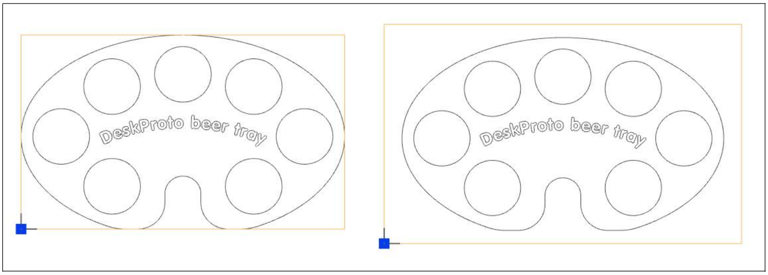
The last option on this page is an important one: the **Z settings**. Entering incorrect Z-settings may result in damaging the working table of your machine. By default the Z=0 is on the top of the block, and the machining depth thus should not be more than the thickness of your material block (as illustrated above). In case of doubt you can add an extra slab of material below the block to protect the table: such slab is called the “wasteboard”.

The **Machining depth** to be set depends on the thickness of your block. We will for instance be using a sheet of 6 mm thick plywood (1/4”), and suggest a depth of -5.9 mm (0.24”) leaving 0.1 mm (0.01”) clearance above the working table. In case you use a wasteboard the machining depth can be equal to the block thickness (or even a bit more). Be aware that the depth needs to be entered with a minus sign, as it is a Z-value below zero.

The **Free movement height** is the Z-level that will be used for positioning movements (traveling from the first contour line to the second, etc). You can imagine this milling operation as an (ancient) pen-plotter: the free movement height being the pen-up level for positioning movements, and the machining depth being the pen-down level, with the pen touching the paper to draw a line.

After entering these Z-values you can press the Next button to proceed to the third page of this wizard, called Material.

The default block size as said is the bounding box of your drawing. While such block will reduce waste to an absolute minimum, our advise is to use a bit larger block size, as that will add room for some clamps on both sides of the part. The bounding box is 360 x 220 mm (14 x 9 inches), and the actual block that we want to use is 400 x 250 mm, 6 mm thick (16 x 10 inches, 0.25” thick). Set the **Dimensions of material block** to Custom, and enter the correct dimensions for X, Y and Z. We entered 400, 250 and 6 mm (16, 10, 0.25”) for our sheet of plywood.



When you now press the Apply button on this wizard page you will see that the drawing has been updated to show the new block size. The beer tray is nicely positioned in the center of the block, leaving ample room for clamps on both sides.

For the **Workpiece zero point** we used the default choice: the front-left corner of the block (so all curves will only have positive X and Y coordinates). If you prefer you can use one of the other options (not available in the Free edition and the Entry edition), like the center of the block. In fact it does not matter what you select here, as long as you use the same location to set the zero point on the machine (more about that later). For the Z no options are available: in vector machining the Z=0.0 always is at the top of the block.

Pressing next will take you to the fourth page of this wizard, called Milling. Here you can set the actual milling parameters and estimate how long it will take to machine the part.

Select a suitable **Cutter** from the list (drop-down menu). Suitable for this project would be an end-mill, so a cutter with a flat tip. The size must be logical for this job: a diameter of say between 1/3 of the material thickness and the full thickness would be nice. This is not critical though, and will also depend on which cutters you have available. When you cannot find your cutter in the list you can easily create a new cutter definition: press button **Cutter library**, OK on warning, button Add, enter all relevant data. Use the Help button for Help. We have used the default cutter (diameter 6 mm, or 1/4” for inch users).

Two different **Speeds** can be set:
The **Feedrate** is the speed with which the cutter travels through the material. For non-metals in most cases the default speed for your machine will be OK, in case you are not sure set a lower feedrate for the first job and try a higher value later.
The **Spindlespeed** is the rotation speed of the cutter. For many small machines this cannot be set from the PC and therefore will have been grayed out in DeskProto. If your machine accepts spindlespeed commands then also for this

setting the default will be a good value to start with.
In general: the smaller the cutter's diameter, the higher the spindlespeed that is needed to achieve the same actual cutting speed (the speed at which the cutting edge moves through the material).

For vector machining the wizard offers two **Toolpath types**:
Profiling will make the cutter follow the curves in the drawing,
Pocketing will make the cutter remove all material inside closed contours of the drawing (this type will be used later for the text).
The third toolpath type, Drilling, is not available via the wizard.
In the Free edition of DeskProto only Profiling is available.

For this beer tray we will apply Profiling: as the cutter will cut on (almost) full depth there is no need to machine the complete area inside each circle as the material within the circles will fall out anyway.
So for Toolpath type select Profiling, and then in the Drop down menu select strategy **Outside/left**. As a result the cutter will travel on the *outside of the outer contour*, as shown by the icon image on the wizard page. The distance between the line in the original drawing (gray) and the toolpath line (red) is called the **cutter radius compensation**: a name that will be self-explanatory.

DeskProto is smart enough to realize that for all closed contours inside this outline the cutter must travel on the other side of the contour-line (the inside). So the 7 round holes in the tray will be machined exactly on the correct size. When you press the button **Calculate** (bottom of the wizard page) you can see the resulting toolpaths, displayed as red lines. Feel free to experiment with the various settings to see what happens.

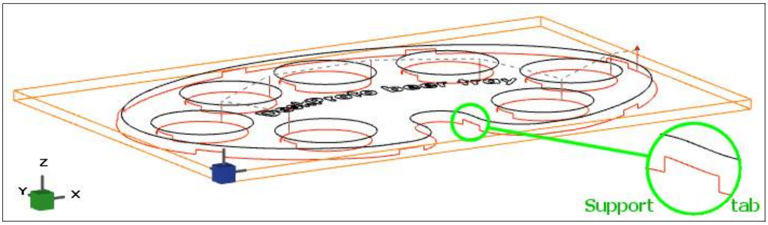
The wizard will apply these settings to **all curves** in the drawing: the outer contour, the seven holes and also to the text. So DeskProto will also try to machine the inside of each character. In most cases this will not be a problem: your cutter will simply be too thick to fit inside these contour lines, and thus they will be skipped. This is true for our 6 mm cutter and also for a 5 mm cutter. Only for a diameter of 4 mm or smaller unwanted toolpaths will be added.

> *In case this happens (if not then ignore this paragraph):*
> *After completing the wizard, open the Vector Operation parameters by double-clicking that line in the tree, go to tab Profiling, and under Select Curves select Custom instead of All (as set by the wizard).*

Now use the button Select to select only the outer contour and the 7 circles: first click outside the part to de-select all, then with the Shift button pressed click on each of these 8 curves. Light gray means not selected, dark gray means selected). After pressing OK twice you will see the updated toolpaths: again save these in an NC program file (Create menu).

Support tabs are interruptions in the toolpath. At these locations not all material will be removed, the remaining material acting as a bridge to keep the part connected to the rest of the material block. This will prevent the part from being damaged when completely cut loose. After finishing all toolpaths you can take the block off the machine and manually remove all tabs. For this project is is OK to use the **Default** tabs. Default tab size is twice the cutter diameter, and default thickness is 50% of the material height. Default distance between tabs is 20 times the cutter diameter.

Roughing (not available in the Free edition) may be needed when the block is too thick for the cutter to remove all material at once. The solution is to use a number of roughing layers: machine the same paths a number of times, each time a bit deeper, until the prescribed depth has been achieved. So check **Use layers** and next a custom **Layer height**. We used 3 mm (0.125”), for our 6 mm (0.25”) thick material this resulted in 2 layers.



Again you can use the Calculate button to see the result. The estimated machining time that is shown will automatically be updated as well. The red lines are the toolpaths, and the dashed lines the positioning movements. The image above shows toolpaths without roughing layers, for a better visibility of the support tabs.

Once more pressing the Next button will open the 5th and last page of this wizard, called Send to machine. As you already calculated and drawn the toolpaths it is not needed to press button **Show toolpaths**. Pressing will in fact make the toolpaths invisible, as this is a “toggle button”.

Button **Show simulation** will make DeskProto calculate and show a simulation: a drawing of the resulting part after machining the toolpaths. The material block, the grooves made by the cutter and the support tabs are clearly visible, making it easy to check if this is indeed the desired part.

For some machines a button **Send to machine** is shown, to directly send the toolpaths to the machine. It is available only on a few machine that support this feature.

If all is OK one more thing needs to be done on this page: pressing button **Write NC-Program file** will open a standard Save-As dialog where you can enter the file-name. The file-extension that is shown (in “Save as type:“) depends on which machine you have selected. Every machine manufacturer prefers some different file format: this is the format suited for the machine that you selected. After that press the **Finish** button to close the wizard.

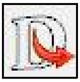
From here you can either read the next half of this lesson, about how to use the dialog-based interface, or jump to the paragraph called “To the milling machine” at the end of this Chapter. The second half of this lesson will include instructions to also machine the text on the beer tray.

Lesson 1B

The Beer tray, dialog based interface

Start DeskProto (or restart it), in the Start Screen (see page 22) again check **Use samples folder** and now select the option **Vector project** (one of the options in 'Start new project'). This lesson will show you how to set all parameters in the Dialog-based interface. Any setting made by the Wizards (as in Lesson 1A) can also be made in this way.

Load the Vector file



The first thing you need to do for a new vector project is load the drawing that you want to use. In most cases this will be an DXF file. In DeskProto you can do so using the command **Load vector file** (located in the File menu), or using the Load vector file button (the 4th button in the toolbar). The result will be a File-Open dialog in which you can browse the DXF file (or AI or EPS) you want to use.

As you selected 'Vector project' in the Start Screen, DeskProto will automatically have opened this Vector-data File-Open dialog for you. Open sample file *2D_DpBeerTray.dxf* - or in case you work in inches file *2D_DpBeerTray_inch.dxf*

You now need to set all parameters for this project, without the guidance offered by the wizard in Lesson 1A. This Tutorial lesson will explain how.
In DeskProto you can enter parameters on three levels:
1- the **Project parameters** are general settings for the complete project, like the machine and the CAD data files that are used. Each project contains one or more parts, for instance two separate model halves.
2- the **Part parameters** define what exactly will be machined: scaling, orientation, partial machining, etc. Each part contains one or more operations, for instance for roughing, finishing, and some detailing operations.
3- the **Operation parameters** define how the part will be machined: which cutter, strategy, speeds, etc. In DeskProto three different operation types are available: Vector operations, Geometry operations and Bitmap operations.

The **project tree** on the left of the DeskProto screen shows this tree structure: one project, containing one or more parts, each containing one or more operations (in the DeskProto Free edition only one part and one operation can be used). You can open the relevant dialog with parameters by double-clicking its line in the tree (or via the Parameters menu).
As you selected 'Vector project' in the Start Screen, DeskProto automatically

created a part with a Vector operation.

In the project tree the first line (the project line) still is called “Untitled”. The project will receive a name when you save the project: the name of the DeskProto project file (.DPJ file) also is the name of the project in the tree. For this lesson the parameters on project level do not need to be changed – assuming that the correct machine has been selected as default machine. You can check that by making the project line in the tree active (clicking on it once): the name of the machine then will be shown on the status bar (bottom border) of the DeskProto window.

The Part parameters

Open the Part parameters dialog by double-clicking its line in the project tree (or via the Parameters menu). Only the tabs that are relevant for a Vector project (called the Vector Settings) are shown, the number of tabs is different for the various editions of DeskProto. Only a few changes are needed: in DeskProto every parameter has a suitable default value, and thus it can be ignored unless a change is explicitly required.

Scaling or Rotating the drawing can be done on the **Transform** tab (in case needed, see lesson 1A). Mirroring is possible too, panning is not useful right now as only one DXF file has been opened.

On the **Material** tab you need to define the size of the material block. Just as in Lesson 1A we propose to use a 6 mm thick plywood sheet of 400 x 250 mm (in inches 16 x 10, ¼” thick). Defining this block in the dialog is a bit more complicated than in the Wizard, as the wizard automatically centers the drawing in the block and the dialog does not. The material tab now shows “*Use all CAD data*”, with the following boundaries:

mm

0.00	360.00
0.00	220.00
-10.00	0.00

inch

0.0000	14.0000
0.0000	9.0000
-0.5000	0.0000

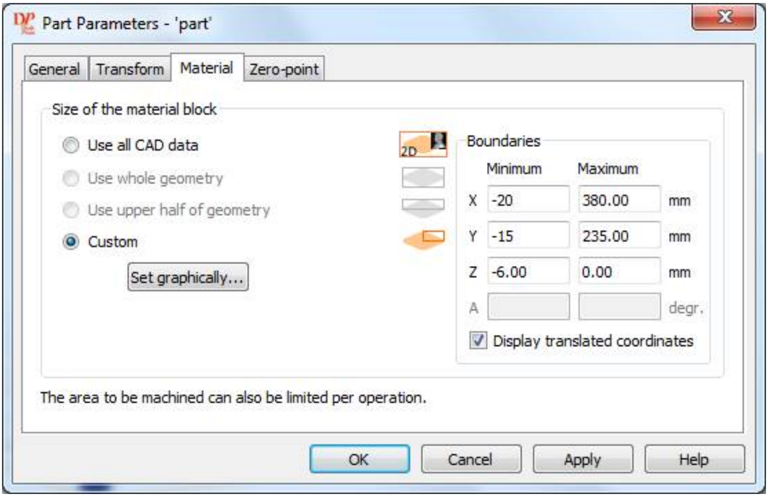
In order to change that while keeping the drawing centered (for X and Y) it will be needed to add the same extra on **both sides**: for X 20 mm (1”) extra for each side, and for Y 15 mm (0.5”) extra.
Select “*Custom*”, and enter the following values:

mm

-20.00	380.00
-15.00	235.00
-6.00	0.00

inch

-1.0000	15.0000
-0.5000	9.5000
-0.2500	0.0000



The minus 6.00 for Z is the thickness of the material that we want to use: enter a different value if you have a thicker or thinner slab of material. When you now press Apply you will see that these values will change: X then will be from 0.0 to 400.0 and Y from 0.0 to 250.0

This happens because the **Zero-point** (the fourth tab of this dialog, not available in the Free edition and the Entry edition) by default is set on the left-front corner of the material block. The drawing clearly shows what has happened: the material block now has the correct size, the drawing is nicely centered within the block and the zero point is at the top-front-left corner of the block.

The Vector operation parameters

In the project tree you will see one vector operation, and its icon shows a red warning sign to indicate that the settings currently are invalid. To see what causes this problem you can open the operation parameters by double-clicking its line in the tree and then again close it by pressing OK: an error message will pop up telling you that “No curves have been selected”. The wizard automatically selected all curves, in the dialog interface this needs to be done by the user, which makes it more flexible.

This flexibility is needed right away, as in this lesson different settings will be used for the contours and for the text. For this first operation only the contours (outside shape and 7 holes) will be selected. Open the operation parameters again, tab Profiling. For ‘Select curves’ choose Custom and press button Select: the dialog “Edit curve selection” will pop up, showing a top view of the drawing.

Click on the outer contour to select it (the color will change from gray to blue). Next with the shift key (on your keyboard) pressed down click on each of the 7 large circles to add them to the selection. As Profiling Strategy select Outside/ Left. When you now press button “Calculate toolpaths” you can check if these are indeed the paths that you need: outside the outer contour, inside all seven circles, and no toolpaths for the text. Close the selection dialog with OK to confirm your selection and press Apply to update the drawing. You will see that the 8 curves you have selected are now drawn in a darker gray.

Other operation parameters need to be set as well:

On tab Profiling select to use the Default **Support tabs**.
On tab page General you can select the **Cutter** to be used and set the **Speeds**. We used a Flat cutter of 6 mm (1/4”) diameter, and the default speeds.
On tab page Z settings you can set the **Machining depth** and the **Free movement height**. We used a depth of -5.9 mm (0.24”) for our 6 mm thick material, and the default free movement height (5 mm).
Finally on tab page Roughing (not available in the Free edition) the option **Use layers** can be checked, with a Custom **Layer height** (we used 3 mm / 0.125”).
For each of these settings a complete explanation can be found in Lesson 1A.
Leave the Operation parameters using OK.

Now you can calculate the toolpaths using this button (or via the Create menu). DeskProto will tell you the estimated machining time (unless you switched that off), and depending on your block thickness and layer height you will see one or more layers of toolpaths: in our case lowering to -5.9 in steps of 3 mm means 2 layers. The current result should be the same as shown at the end of Lesson 1A.

Adding the text to be engraved

The text in the drawing needs to be machined in a completely different way: a smaller cutter, a much smaller machining depth, and all material inside each character needs to be removed. In order to achieve this a second vector operation is needed.

For this engraving toolpath a second operation is used, and Pocketing type toolpaths. As multiple operations and pocketing are not supported in the Free edition *users of the Free edition can skip this paragraph*.

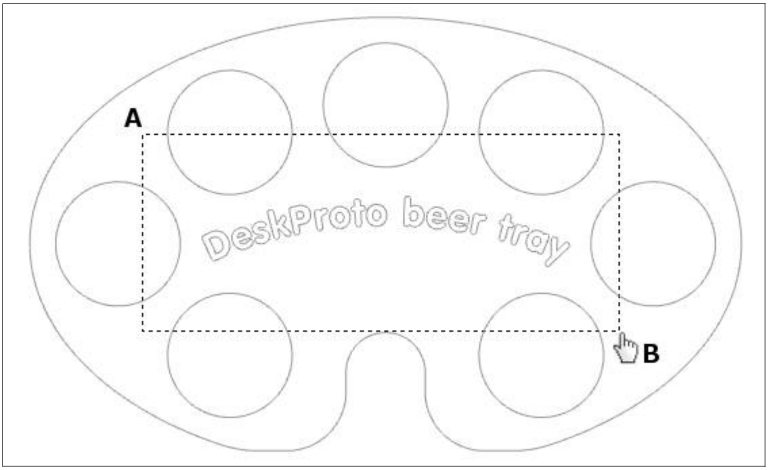
Easiest way to add an operation is to right-click on line Part in the tree, and then in the context menu select **Add Vector Operation**. The new line will display a red warning sign in its icon, caused by the same error as for the first operation: no curves have been selected yet.

Open the vector operation parameters for this new operation. The settings to be changed will be listed below, with a short comment only.

Tab **General**:
- Change the Name from “vector operation [#1]” to “Text”: clear naming will make things easier later.
- Select a thinner cutter: a flat tip of 2 mm (1/16”) diameter will (just) fit inside each character.
- For this thinner cutter set a lower feedrate and if possible a higher spindle speed.

Tab **Z settings**:
- Set the machining depth on -0.5 mm (-0.02”): enough to engrave a readable text.

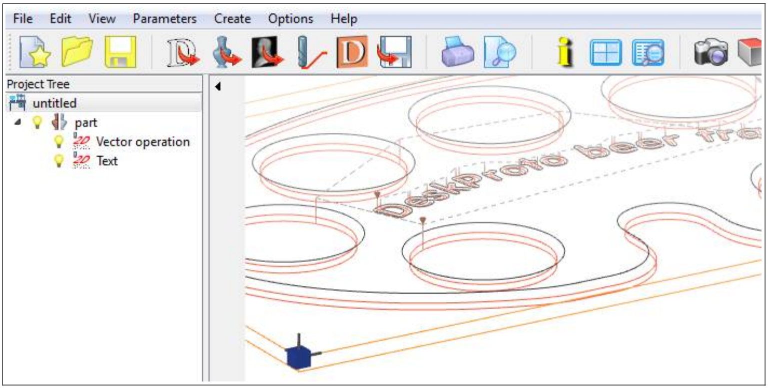
Tab **Pocketing**:
- Removing all material within a closed contour line is called Pocketing. So on that tab set the Curve selection to Custom and press the Select button.



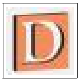
Now in the Edit curve selection dialog select the text curves (26 curves).When selecting you can use a 'rubberband' selection tool: position the cursor close to point **A** (make sure that none of the curves is activated (purple), press the left mouse button, move to point **B** and release the mouse button. Now only the curves that are completely within the dashed rectangle (the rubber band) will have been selected: 26 curves.
As **Strategy** select Offset, the default Stepover of 50% is OK.

Again press button Calculate toolpaths to see the result: toolpaths to remove all material inside each of the text curves. As you can see DeskProto automatically recognizes that curves within other curves need to be treated as islands without toolpaths.

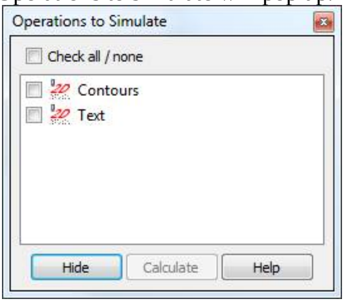
The dashed lines in gray are positioning movements on Free movement height.
After closing both dialogs and again pressing Calculate toolpaths the DeskProto screen now shows the toolpaths for both operations:



The first operation has not yet been renamed, we advise to rename it now as the use of proper names will prove to be convenient when reusing the project later. Also the name of the project (the upper line in the tree) still is “untitled” as you have not yet saved the project.



The next step is optional: you can show a simulation of the resulting part. When you press this button the material block will be shown on your screen as a solid, brown block (in most cases hiding the CAD data and the toolpaths), and a dialog called Operations to simulate will pop up. In this dialog all operation in this part are shown, each with a checkbox to select it. Check the first operation (the Profile toolpaths) and press Calculate: the simulation will be updated to include the profiling toolpaths. You can include the Text Operation by also checking that and again pressing Calculate.
The level of detail of the simulation can be set in the part parameters (tab Simulation).



Finally you can write the NC file, either via the Create menu or using the button. For most machines you will see that two different NC files are written: after the first operation DeskProto finds that a different cutter is required and thus has to start a new NC file. Only for machines with an Automatic ToolChanger (ATC) both operation can be saved in one NC file.

It is also possible to save these files one by one. In the tree you can see a yellow light bulb in front of each operation. Clicking on such light bulb will make it gray, and the operation becomes invisible. That is a very handy feature when working with several operations. In case not all operations are visible when you tell DeskProto to write the NC file, DeskProto will ask “Do you want to use the visible operations only?” This makes it possible to save the NC files one by one, giving a clear and informative name to each file.

From here you can proceed to paragraph “To the milling machine”, which will explain about this next step.

To the milling machine

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

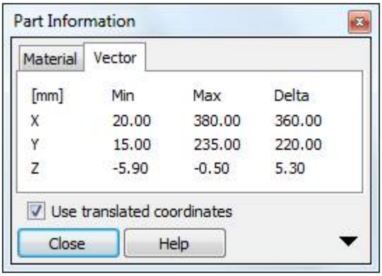
First the block of material has to be prepared, for which you have just entered the dimensions (in one of the previous two paragraphs). For our beer-tray sample we used: 400 x 250 x 6 mm, or in inches 16 x 10 x 0.25” - of course you may have a different thickness or size. For X and Y some oversize is not a problem, the Z (material thickness) needs to be exact though.

As this block is larger than the part to be machined you can fixture the block on the machine’s working table using clamps on all four corners: the toolpaths will not come close to these four clamps. As an alternative you can use double-sided adhesive tape. The image above shows the four clamps, and also an extra slab of material below to be used as “wasteboard”: this will protect the machine’s working table in case the cutter travels too deep. This should not happen, still for a first project it is good to be extra careful.

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

In DeskProto you have defined the left-front corner of the block to be (0,0), and for vector machining Z=0 always is at the top of the block. This means that all X and Y positions in the NC file are positive (X=0 is the left side of the block, and Y=0 is the front side), all Z-positions (except for positioning moves) are negative (Z=0 is the top of the material block). So now on the machine you need to set the WP zero point at the **Left-Front-Top corner** of the material block that you just fixtured on the machine.

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



Finally double-check if the extreme values of the toolpaths: can the machine reach the minimum values for X and Y, and can it travel to the lowest Z-value without causing any damage. You can find these value in DeskProto's Part information dialog, see the image above (make sure to check "Use translated values" to see the same coordinates as in the NC file).

Now you are ready to start the machine by sending the NC program file you just created to the machine. Most CNC milling machines have their own **control software** to do this (like Mach3, PCNC, LinuxCNC, ...). If so then exit DeskProto, start this machine-control program and open the NC program file. If needed first transfer this file from the DeskProto PC to the machine's control PC. The command to start machining can be given in the control software.

A few 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 one of the options 'Send 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). The Send to machine option is not present in the versions for MacOS and for Linux.

In short:
- Clamp the material on the machine's working table
- Set the WorkPiece zero point on this block as defined in DeskProto
- Load the NC file from DeskProto
- check if the extreme values of the toolpaths fit
- Start the machining process.

For the second NC file, with the text, this process can be repeated.
Leave the material clamped as it was and mount the smaller cutter. The X=0 and Y=0 positions remain the same as for the first NC file. Only the Z=0 will need to be set again, now with the tip of this new cutter touching the top of the block. After you have done so:
- Load the second NC file from DeskProto
- check if the extreme values of the toolpaths fit
- Start the machining process.

At the end of the milling process the model will still be attached to the remaining block of material by means of the support tabs. You can manually remove the beer-tray from the block and clear the seven holes. Finally use some sanding paper to remove the remains of the support tabs and smoothen all contours.

2. Picture Frame (basic geometry)

Lesson Two

In this second lesson you will learn how to machine a 3D geometry. Again some DeskProto basics will be explained, partly repeating Lesson One. This time a geometry file will be processed and a 3D NC file will be made, ready to send to the milling machine. The lesson again will be presented **twice**:
first using the **Wizard 'Basic Geometry machining'** and next using the **Dialog-based interface.** This lesson is for all DeskProto editions.

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

Start DeskProto

Start DeskProto, as explained in Lesson One, and proceed until the Start screen is displayed. In case this screen does not pop up automatically you can open it in the File Menu.

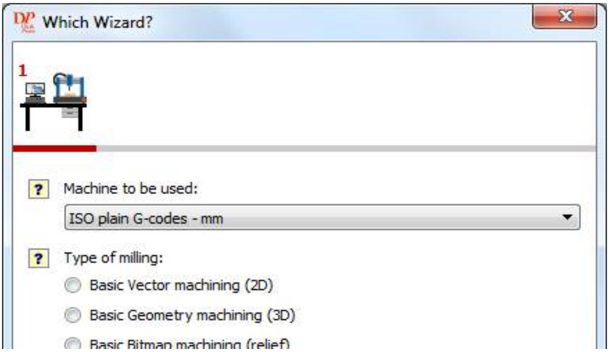
In this second lesson again both user-interfaces that DeskProto offers will be explained: the Wizard-based interface in **Lesson 2A** and the Dialog-based interface in **Lesson 2B**. Both versions of this lesson will lead to the same result.

In order to start Lesson 2A: in the Start Screen please check **Use samples folder** and then select the option **Use wizard**.

Lesson 2A

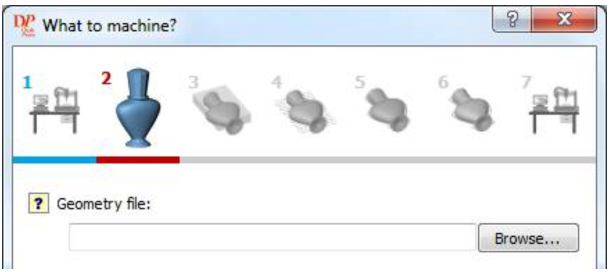
The Picture Frame, wizard interface

The DeskProto **Wizard interface** makes the program very easy to use for anyone without previous experience. We will keep the Tutorial as short as possible, as the wizard should in fact be self explanatory...



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

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



Just as in Lesson One the second icon becomes active (enlarged and underlined in red): you are on the second page of this wizard. You can see that this wizard has more pages than the wizard in Lesson One, and that the navigation icons are different. When using the DeskProto **Expert** Edition or **Multi-Axis** Edition you will see **seven pages**, as shown in the illustration. When using the **Entry** Edition you will see **six pages**, as a *Contouring* operation is not possible in this edition, while in the Free edition only five pages are present (no *Roughing*). Nevertheless also in the Entry and the Free edition this lesson can be completed, as these extra operations are not required.

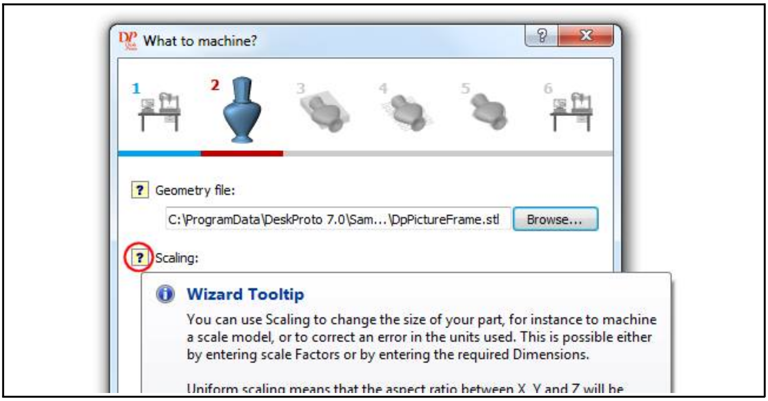
On this second page you need to browse a **Geometry file**. As you started with option "Use samples folder" checked, the Browse button should directly look in the DeskProto Samples folder: select file *DpPictureFrame.stl* and press Open.

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

In case you do not see the Samples folder: you can find it in folder \ProgramData\DeskProto 7.0\Samples\ (for more information see the earlier paragraph on Files and Folders)

Scale when you need a different size picture frame: you can create a frame for any size photo as DeskProto allows "non-uniform" scaling: different for X, Y and Z. Do not change the Orientation as for this frame the Top surface needs to remain on top.

For more information on any of these settings: move the cursor over one of the **yellow question marks** to get an explanation in a "Wizard Tooltip".



On the third page called “Material and Support” no changes are needed for this project: use the geometry's bounding box as material block, do not add supports and keep the default zero point position. These last two options (supports and zero point) are not present for users of a DeskProto Free or Entry Edition.

The next three wizard pages are for the three operations, as this wizard will automatically generate three operations: **Roughing** (optional, not in the Free edition), **Finishing,** and **Contouring** (optional, not in the Free and Entry editions). The roughing operation is meant to quickly remove material, the finishing operation to create an accurate model with a smooth surface, and the final contouring will smoothen out most staircase steps that may have remained along the outside contour, using the strategy “Contour Only”. On these three pages you need to enter the actual Milling parameters like cutter, speeds and precision.
The Free edition will create a project with just one operation (finishing), the Entry Edition will skip contouring and create a project with two operations.

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

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

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

The last page of the wizard, **Send to machine**, shows the resulting project tree (you can rename any line in the tree after a slow double click). Three or four large buttons are shown:
Button **Show Toolpaths** makes the toolpaths visible / invisible, and when needed calculates the toolpaths.
Button **Show Simulation** does the same for the Simulation, and for a clear view switches off the toolpaths when visible.
Button **Send to machine** is available only when your machine supports that option (very few machines do) and when an NC output device has been configured in the DeskProto preferences.
Button **Write NC program file** in most cases is the last step to be done in DeskProto. This NC file then can be sent to your CNC milling machine, see the end of this Chapter. The file extension depends on the machine that you have selected: each machine manufacturer uses a different type of NC file.

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

Before you start machining please read the below **Notes** about this result:

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

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

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

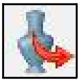
From here you can either read the next half of this lesson, about how to use the dialog-based interface, or jump to the paragraph called “To the milling machine” at the end of this Chapter.

Lesson 2B

The Picture Frame, dialog-based interface

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

Load the Geometry file



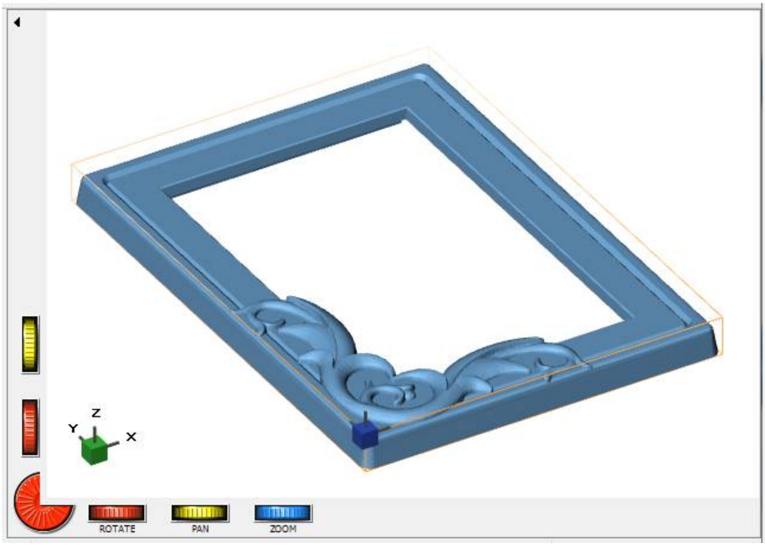
The first thing you need to do for a new project is load the geometry that you want to use. In most cases this will be an STL file. In DeskProto you can do so using the command **Load geometry file**, located in the File menu, or using the Load geometry button. The result will be a File-Open dialog in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

The file that we want to use for this lesson is called **DpPictureFrame.stl** and is located in the DeskProto Samples folder. Unfortunately the location of this folder – the Windows \ProgramData\ – is different per Windows version, so the file may be difficult to find. No problem: you can use the checkbox in the Start Screen mentioned above. So please go to the Samples folder and open the file DpPictureFrame.stl

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

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: a nicely decorated photo frame. The result of this lesson will be a great product, either on your own desk or to be used as a self-made gift !

The light-brown lines around the geometry show the bounding box: the dimensions of the material block that is needed to create this part. You can make it more clearly visible by double-clicking on the image and in the **Items visible** dialog (explained below) checking the option Translucent for Material block. You can restore the previous image by again un-checking this option, as you prefer.
The **green cube** with X,Y and Z shows the directions of these three axes, and is called the Orientator. The small **blue cube** shows the position of the WorkPiece zero point (0,0,0) for this part, more about that will follow later.

Rotate, Pan and Zoom

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

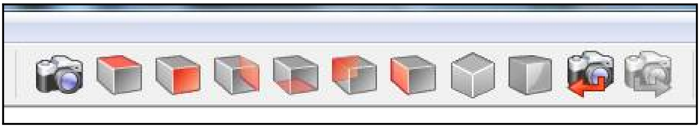
The vertical and horizontal red thumb-wheels offer rotation, around a horizontal and a vertical axis (horizontal and vertical on your display screen).

The three-quarter-round wheel in the corner allows you to rotate around the axis perpendicular to the screen. Position the cursor over one of these red thumb-wheels, press the left mouse button and keep it pressed while moving the mouse. You will see that the geometry rotates as indicated.
The two yellow thumb-wheels are for panning (moving the geometry over the screen, horizontally and vertically). And the blue thumb-wheel is meant for zooming (changing the viewing-distance).

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

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

It is even possible to rotate, pan and zoom without using these mouse function buttons: press the middle mouse button (the mouse-wheel) for panning, and rotate this wheel for zooming. When zooming the cursor position sets the center of the zoom, so you can zoom onto any detail on the screen.
DeskProto supports the SpaceMouse ® by 3Dconnexion for rotating, panning and zooming.

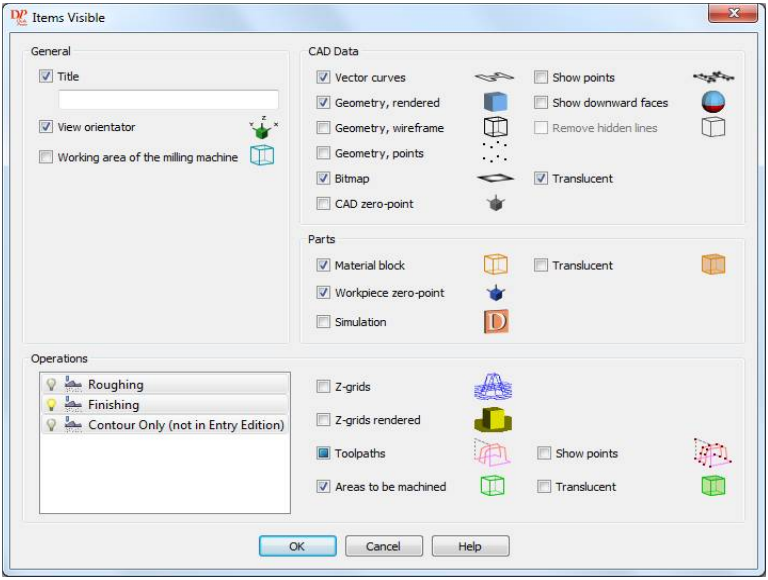


A number of standard views can be set very quickly using the eight toolbar buttons showing small cubes. Each colored button sets a **main view** (a view along one of the main axes). The next four buttons to the right of these cubes can be used to quickly set isometric view, default view, previous view and next view (the latter only enabled when button previous view has been used).

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

Items visible

This option - that will prove to be very handy - allows you to select which items should be displayed in the View Window. You can open the Items visible dialog by clicking this command in the View menu or by double-clicking inside the view window.



Every displayable item is shown, with a checkbox to mark for each item whether or not it should be displayed. When checked (“V”) the corresponding item will be displayed after pressing the OK button.
When the checkbox is completely filled (showing a square instead of a V), like in the illustration above for the Toolpaths, it means that multiple operations are selected (light blue background) having a different status for that item.

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

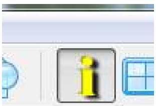
Check the model’s orientation and dimensions

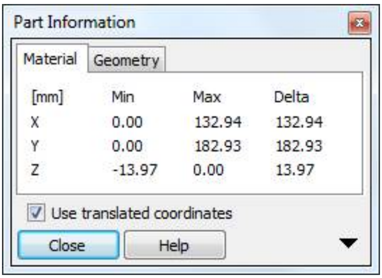
While examining the geometry by looking from various directions, you will have noticed that the Picture Frame is correctly positioned for milling. In DeskProto the milling tool always comes from the positive Z-direction: X is from left to right on the milling machine, Y is from front to back, and Z is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

Two features for checking the orientation have already been mentioned:
- the Orientator (green cube with three axes) shows the direction of the axes
- the option Show downward faces can be used to check for undercuts.
This geometry does not need to be rotated.

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

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





Press the Toolbar button with the yellow **i** or select Part Information from the View Menu in order to display the Part Information dialog: see the images above. This dialog gives you the dimensions both of the material block that you defined and of the geometry that is used for this part (after scaling, rotating and mirroring). When “Use translated coordinates” is checked the coordinates shown will be the same as used in the NC file. For your current settings both tab pages will show the same result.

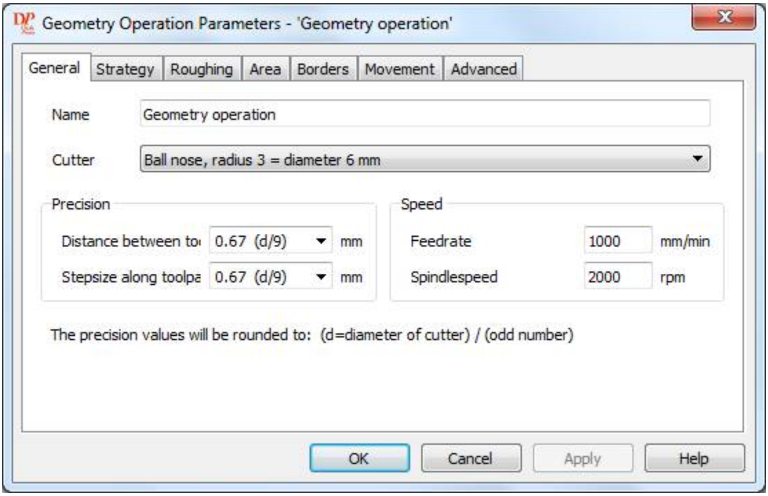
As you can see the dimensions of the part are OK (the illustration is in mm), so you can proceed and set the milling parameters.

Set the milling parameters

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

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

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



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

The Operation Parameters dialog consists of a number of Tab screens (in the Free edition and the Entry edition less tabs are present than shown above). As all milling parameters have suitable default values, and as we want to start simple: for now only look at the front Tab page (“General”) and just ignore the hidden Tab screens. Do not ignore the most important button in the dialog: the **Help** button. It will lead to a page that completely explains this one dialog. Please try the Help, and remember this for when you have any question later.

The Operation parameters that you see now are different from the dialog that you saw in Lesson one: then you saw the Vector operation parameters, now the Geometry operation parameters. In addition DeskProto includes a third operation type: the Bitmap operation, to be covered in the next lesson.

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

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

Smaller precision values will lead to a smoother and more accurate result, however also to a longer machining time. DeskProto will automatically show you the estimated machining time after calculating the toolpaths.

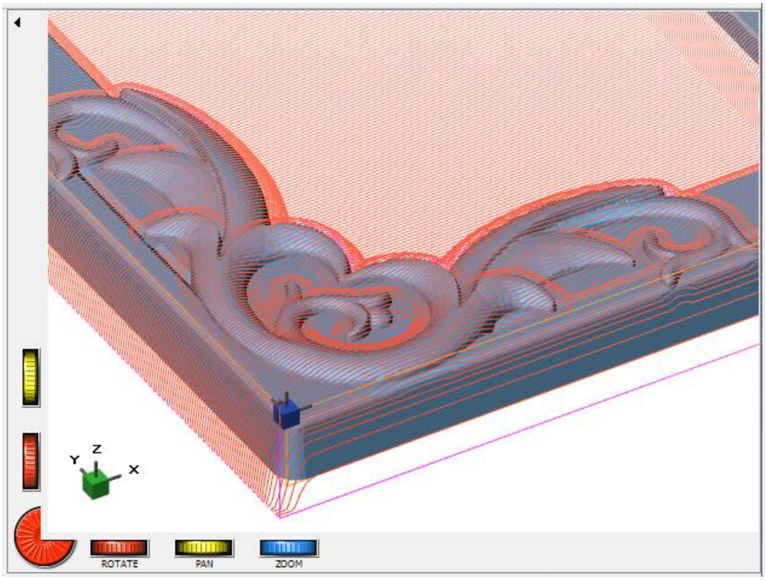
For machining foam, tooling board and wood the default values for **Feedrate** and **Spindle speed** will be OK; generally speaking these need to be changed for harder materials only. So now press OK to close the Operation Parameters.

Calculate the Toolpaths

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

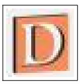
During the calculations DeskProto shows a progress-bar to keep you informed about their progress.

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



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

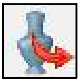
Show a Simulation



The next step is optional: you can show a simulation of the resulting part. When you press this button the material block will be shown on your screen as a solid, brown block (in most cases hiding the CAD data and the toolpaths), and a dialog called Operations to simulate will pop up, as explained in the previous lesson. Check the operations that you want to simulate and press Calculate to show the simulation, button Hide will make the simulation invisible.

Create NC program

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


Load geometry


Calculate toolpaths


Write NC-file

You now have seen that these buttons (nrs 4/5/6 for Load, 7 for Calculate and 8 for Write NC) are the central buttons in the DeskProto work flow for 3D Geometry machining.

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

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

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

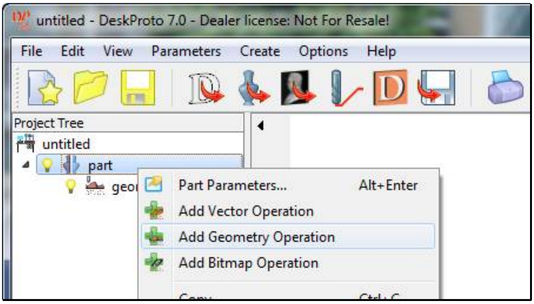
From here you can either proceed to paragraph “To the milling machine”, or first learn about roughing and finishing in the next paragraph.

Roughing and Finishing

The toolpaths that you just made will indeed produce a picture frame: as the part height is small it can be machined directly at full depth. So you are free to skip the fine-tuning that is presented in this paragraph. Still, applying Roughing and Finishing has a few distinct advantages, so it will be worthwhile to bear with us for some more time. This paragraph does not apply to the Free edition as that does not support Roughing.

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

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



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

Double-click on the first operation line in the tree and change its name to “Roughing”. Now you can set the Operation parameters to make this a real

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

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

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

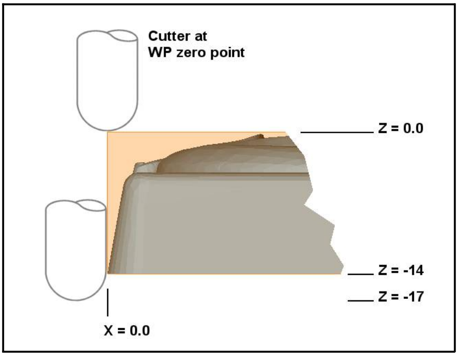
The second operation will be the **Finishing Operation**: open the parameters, and change the name to 'Finishing'. You do not want any Roughing functions active, and you want smaller Precision values (Distance and Stepsize). For finishing we often check the option 'Skip horiz. Ambient' in order to finish only the geometry surface (tab Advanced). For this frame also uncheck "ignore enclosed ambient"), which will make DeskProto skip the large center hole. Make sure that on tab Movement the option Sort has been checked, as otherwise many unneeded positioning movements will result. You can leave all other settings as they were (default values).

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

To the milling machine

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

First a block of material has to be prepared. You already know the dimensions of the frame as you have just checked these in the Part Information dialog:
133 x 183 x 14 mm, or in inches 5.23” x 7.2” x 0.55. The values in the dialog are in fact a tiny bit smaller, which you can ignore for this project.



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

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

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

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

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

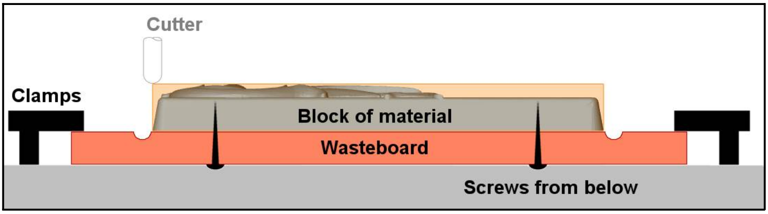
On many machines you can enter the WP zero point by manually positioning the cutter (milling tool) exactly on the desired workpiece zero point, and then telling the machine controller that this is position (0,0,0). Keep in mind: for X and Y the center of the tool must be positioned, for Z the tip of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

Now you are ready to start the machine by sending the NC program file you just created to the machine. Most CNC milling machines have their own **control software** to do this (like Mach3, PCNC, LinuxCNC, ...). If so then exit DeskProto, start this machine-control program and open the NC program file. If needed first transfer this file from the DeskProto PC to the machine's control PC.
Some 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). The Send to machine option is not present in the versions for MacOS and for Linux.

At the end of the milling process the model will still be attached to the remaining block of material, as for this first test your block was larger than the model, and as a 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 rest of the block using for instance a small band saw machine.

For **the definitive model**, so a frame without any excess material, the block needs to be more accurate and the fixturing more precise. Make the block some millimeters larger for both X and Y, to compensate for possible positioning errors (not too large as then the chips cannot easily fall off during machining). Make the Z (thickness of the block) as exact as you can. Setting the WP zero point now needs to be done accurately, exactly at the Top Front Left corner of the block. This will be easy, as the process is the same as for the test just done.

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



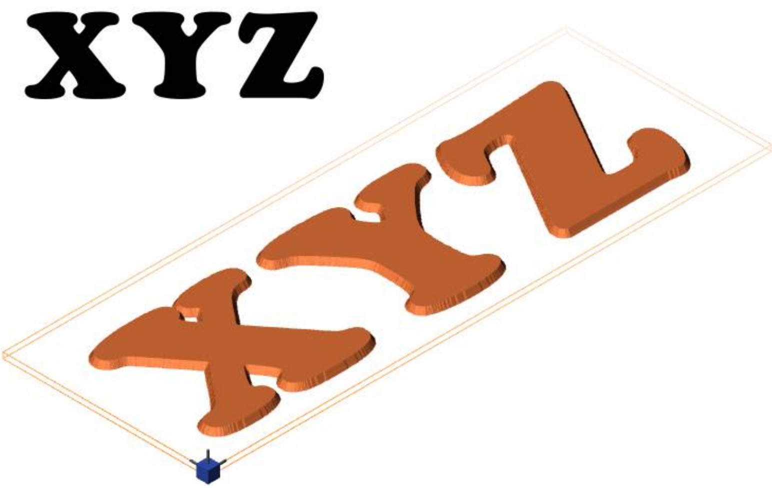
The screws method works out nicely: see the illustration above. You can safely use clamps to securely fixture the wasteboard as the cutter will not come near these clamps. You only need to take care to correctly position the screws: they may not connect to a part of the block that will be milled off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

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

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

3. XYZ logo (basic bitmap)

Lesson Three



This third lesson you will teach you the basics of DeskProto bitmap machining. Again some DeskProto basics will be explained, partly repeating the previous lessons as that will allow you to start reading here. A simple bitmap file will be converted to a 3D relief and a 3D NC file will be made, ready to send to the milling machine. The lesson again will be presented **twice**:
first using the **Wizard 'Basic Bitmap machining'** and next using the **Dialog-based interface.** This lesson is for all DeskProto editions.

The bitmap image and the relief are both shown in the figure above: it is the **company logo** for the (imaginary) company named "**XYZ**". Bitmap machining may not be the optimal way to create this relief, still it offers a very easy way to do so. In case you only have the graphics as bitmap and do not know how to convert it to vector data this lesson will show you what to do to get the project done.

Start DeskProto

Start DeskProto, as explained in Lesson One, and proceed until the Start screen is displayed. In case this screen does not pop up automatically you can open it in the File Menu.

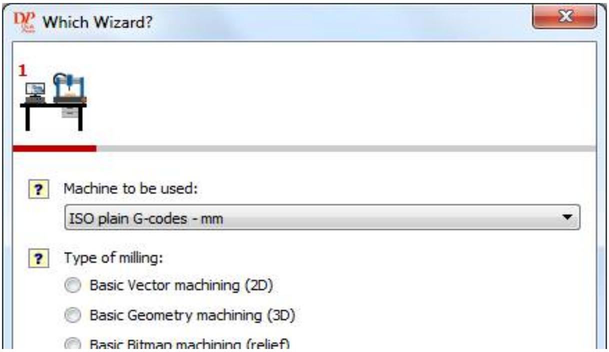
In this third lesson again both user-interfaces that DeskProto offers will be explained: the Wizard-based interface in **Lesson 3A** and the Dialog-based interface in **Lesson 3B**. Both versions of this lesson will lead to the same result.

In order to start Lesson 3A: in the Start Screen please check **Use samples folder** and then select the option **Use wizard**.

Lesson 3A

The XYZ logo, wizard interface

The DeskProto **Wizard interface** makes the program very easy to use for anyone without previous experience. We will keep the Tutorial as short as possible, as the wizard should in fact be self explanatory...

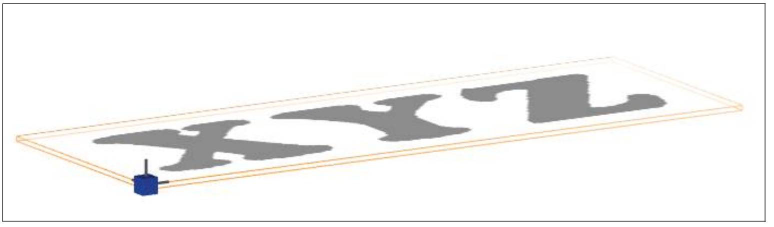


The Machine to be used should already be the correct machine, as you have set the default machine when first starting DeskProto. If not correct you can select a different machine here (changing the default machine can be done in the Default Part parameters (Options menu).

In fact a series of different wizards is available, each meant for a specific type of milling. For the XYZ company logo we will use the third wizard:
Basic Bitmap machining, that is available in all Editions of DeskProto. So please select that wizard and press Next.

The second icon becomes active (enlarged and underlined in red): you are on the second page of this wizard. You can see that this wizard offers five pages (in the Free edition four). This second page offers all setting to define what to machine.

Obviously the first step is to **load the bitmap file** to be used: press the Browse button to open a file. As when starting the wizard you checked ‘Use samples folder‘ this is the folder that will show; it will include the sample file **XYZlogo.png** that we used for this lesson. So open this file (you may of course also use your own bitmap file).



The file will be shown on the screen immediately. Do no worry about the image being rather vague and having a blurred outline: DeskProto shows a simplified version of the image in order to speed up the (3D) graphics. For toolpath calculations the real image will be used. A color image will be automatically converted to gray values by DeskProto.
The **orange lines** show the block, that at this point matches the bitmap size using the default settings for scale and for Z. You can also see the block size in the Part information dialog that popped up.
The **blue cube** shows the position of the zero point: the left-front corner, at the top of the block. You will later need to set that same zero point on your machine.

The image size for our file XYZlogo.png is 279.4 x 101.6 mm, or for inch users 11 x 4 inch.
For readers who want the exact information: the bitmap file is sized 3300 x 1200 pixels, at 300 DPI (Dots Per Inch). You can check this in DeskProto’s Project parameters dialog: tab Bitmap, button File info...

The **Scaling** option in this wizard allows you to set the size of the relief to be machined. Easiest is to select “Dimensions” and then enter the required size. We want to make a small name-tag, so we enter 100 mm as X-size (that equals about 4 inch). You will see that the Y-size automatically follows and changes to 36.36 (ca 1.45”): this happens because the checkbox “Uniform” has been checked, so the same scaling factor is applied to both axes.
When pressing the Apply button (bottom left corner of the wizard) the image and the Part Information dialog are updated to reflect this new size.

The thickness of the relief is set by the **Z-settings**: you can enter a Z-level for (pure) black and one for (pure) white. All intermediate gray values will be automatically calculated. For our small name tag a relief depth of 1 mm is sufficient. We want our logo on top of the surface (embossed characters), so black must have the highest value: Z=0 and white must have the lowest: Z=-1. This will make DeskProto remove all material around the black logo. Positive Z-values are not permitted here as the top of the material block is on Z=0. So enter -1.0 mm (0.04”) for white and 0.0 for black

You now have defined *what to machine* (the Part). Press Next to continue and define *how to machine* it (the Operations).

Wizard page 3 (not present in the Free edition) can be skipped: for this small relief depth (1 mm) a **Roughing** operation is not needed. So simply un-check the box “Use roughing operation” and again press Next.

Wizard page 4 defines the **Finishing** operation, and cannot be skipped.

For an embossed logo the ideal **Cutter** is a conical one with a flat tip. Such cutter is also called “V-shape cutter”, as it’s shape is a V. For our project we selected the cutter called “*Conic engraving tool 30 degrees*” from the drop-down list. This cutter has a side angle of 30 degrees, which will result in a 30 degree surface at the logo’s outer contour. Also a flat tip of 0.2 mm (0.008”) diameter is present, which makes it possible to machine a perfectly flat surface around the logo.

Note the difference between the “grinding angle” (as shown in the image) and the “included angle” that is used by some cutter suppliers. For this cutter the included angle (top angle) is 60 degrees.

In case the cutter that you have is not present in the drop-down selection list you can easily add is to the list after entering the **Cutter library**. After pressing OK for the warning press Add in the library and enter all cutter details. Use the Help button for Help. In the Cutter library you can also press Edit to check (and/or to edit) the exact cutter definition for one of the existing cutters.

Next setting is the **Distance between toolpaths**. As we just selected a conical cutter this setting needs some extra care: the distance to be selected must be smaller than the tip diameter of the cutter. If not than a small ridge of material (a “cusp”) will remain between each two toolpaths. The tip diameter of our cutter was 0.2 mm (0.008”), so we selected toolpath distance 0.18 mm

(0.0072”). The smaller this distance, the smoother the result. However of course the machining time will increase as well.

When machining in a light material (non-metal) in most cases the default **Speeds** need tot be changed. This case is an exception though, because of the very small tip diameter of this cutter. The diameter of a cutter determines the actual cutting speed (speed of the cutting edge cutting through the material): the smaller this diameter, the smaller the actual cutting speed for the same rotation speed in rpm (Rounds Per Minute). So for this conic cutter it will be best to select a higher Spindlespeed.
As this tip is very small you may also want to set the Feedrate a bit lower than default.

The default **Strategy** (cannot be set in the Free and Entry editions) is Parallel: the first toolpath is at the front side of the part, from left to right (at constant Y), next one step along Y and then from right to left back to X=0. Again a step along Y, etc etc. This is the most simple and straightforward strategy, which will work OK for this name badge.
As an alternative you can select strategy Waterlines: the toolpaths are much more complex (and the calculation will take longer), however the result may be better as the cutter toolpath will follow the outside contour of each character.

Finally press button **Calculate** to make DeskProto calculate and display the toolpaths (you can do this again and again to see the results for each strategy). After calculation DeskProto will also show you the **Estimated machining time** for this operation (if not then you can open this dialog in the Create menu, after finishing the wizard). This is an estimation: the exact machining time is influenced by many factors that DeskProto does not know.

Pressing Next will lead you to the final page of this wizard. Here you can first check the result that you can expect by pressing button **Show Simulation**. The most important button on this page is **Write NC-Program file…**, which will open a Save-as dialog to export the toolpaths to an NC file. The file extension of an NC file differs per machine: DeskProto will automatically use the correct file type for the machine that you selected when starting this wizard. After that press the **Finish** button to close the wizard.

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

Lesson 3B

The XYZ logo, dialog based interface

Start DeskProto (or restart it), in the Start Screen (see the previous lessons) again check **Use samples folder** and now select the option **Bitmap project** (one of the options in 'Start new project'). This lesson will show you how to set all parameters in the Dialog-based interface. Any setting made by the Wizards (as in Lesson 3A) can also be made in this way.

Load the Bitmap file

The first thing you need to do for a new bitmap project is load the image that you want to use. Various image file types are supported: BPM. JPG, GIF, PNG and TIFF. In DeskProto you can do this using the command **Load bitmap file** (located in the File menu), or using the Load bitmap file button (in the button toolbar). The result will be a File-Open dialog in which you can browse the file that you want to use.

As you selected 'Bitmap project' in the Start Screen, DeskProto will automatically have opened this Bitmap-data File-Open dialog for you. Open sample file *XYZlogo.png*
This is a very simple bitmap design, as only two colors are present: black and white, (almost) without any intermediate gray values. So the resulting relief will be simple as well and show just two Z-levels. Which is fine for this lesson about *basic* bitmap machining. As already mentioned before: a low-res version of the bitmap is displayed in order to have fast 3D graphics. The black of the bitmap shows as gray because it is displayed as Translucent: otherwise the toolpaths would later by obscured by the bitmap. You can switch off this Translucency in the Items Visible dialog (doubleclick on the image to open that dialog).

You now need to set all parameters for this project, without the guidance offered by the wizard in Lesson 3A. This Tutorial lesson will explain how.

First the Part parameters will be done: defining the Part that we want to machine. You can open the **Part parameters dialog** by double-clicking in the tree, on the line called “part”. The part parameters that you see now are the settings for Bitmap: almost (but not completely) equal to the Vector settings and the Geometry settings used in the previous two lessons. As always in DeskProto: as all parameters have a suitable default only a few need to be changed for this project.

On tab **XY Transform** you need to set the **Dimensions** of this part. Just as in lesson 1A the default size follows the size as set in the file (at 300 DPI): 279.4 x 101.6 mm (11”x 4”). Many more sizing options are present here than in the wizard. In this lesson we will simply select Custom and enter a size of 100 mm (4”) for X. The Y size will follow as the aspect ratio is kept: 36.36 mm (ca 1.45”).

On tab **Z-Settings** some changes are needed as well, in order to define the relief depth. Just as in lesson 1A we will set -1.0 mm (.04”) for white and 0.0 for black: a flat surface at Z = -1 for the background area, and the three embossed characters 1 mm higher.
The default material block definition (“Use all CAD data”) is OK, and so is the position of the zero point (right-front-top corner of the block). So you can now press OK to close the part parameters.

The next step is defining how you want to machine this part. This can be done in the Operation parameters: in the tree double-click the line “Bitmap operation” to open the **Bitmap Operation Parameters dialog**.
Most settings for this simple project are present on the first tab page (General):

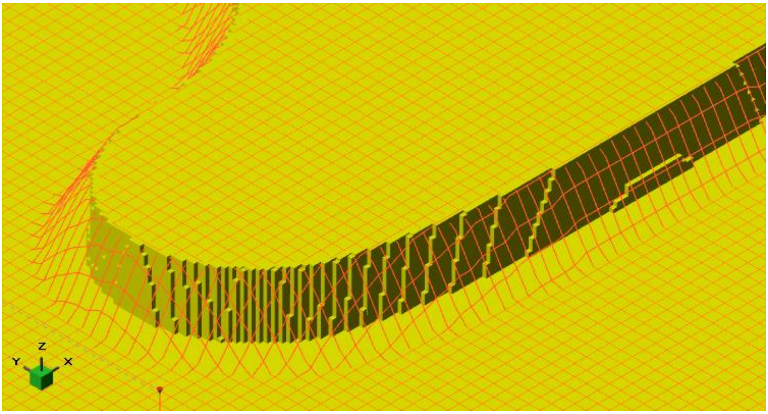
As a **Cutter** we suggest to use a conical cutter: select the cutter called “*Conic engraving tool 30 degrees*” from the drop-down list. Also see the image and the explanation in the previous paragraph.

The **Distance between toolpaths** needs to be smaller than the 0.2 mm tip diameter of this cutter, so again we selected 0.18 mm (0.007”). For the Stepsize we used the same value, which in most cases is a good choice.

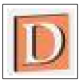
Set the **Feedrate** a bit lower than its default, and **Spindlespeed** a bit higher, because of the very thin tip of the cutter that will do the actual work. This manual cannot give any numbers here as these will be different per machine. The default values that DeskProto shows for your machine will offer a nice guideline.

If you prefer a different **Strategy** you can select one on the Strategy tab: the small icon images show you what each strategy offers. For this name-badge either Parallel (which is the default) or Waterlines will be a good choice.
On the other tab pages no changes are needed, so you can close this dialog and enter your changes by pressing OK Not all tabs are present in the Entry edition, and the Free edition offers only the General tab.

As you now have set all parameters you can proceed and calculate the toolpaths, by pressing this button. DeskProto will display them as red lines. After calculating a dialog will pop up that gives the estimated machining time – if not then you can find this dialog in the Create menu, with an option to “Always show this dialog after calculating toolpaths”.



Some background information:
The image above shows you how DeskProto calculates the relief. Shown are the “Z-Grid” and the toolpaths (strategy Crosswise). The two Z-levels (Z=0 for black and Z=-1 for white) are clearly visible. At the outer contour of this character you can also see grid-cells with an intermediate Z-level. These are present as in the original bitmap pixels with an intermediate gray-level have been added for anti-aliasing (making the outer contour visually smooth).
The toolpaths do not reflect these in-between grid-cells, as the tip of the cutter cannot reach them because of 30 degree V-shape of the cutter. As a result the outer contour will be a nicely sloped surface at 30 degrees.



At this point you can optionally display a Simulation of the resulting part, via Create > Calculate simulation or using the button shown left.

When you are happy with the result you can press button Write NC-file in order to save this complete toolpath in a file. Select a proper Name and Location, allowing you to easily find the file again when you start the machining process.

To the milling machine

The NC program file that you just created is ready to be sent to the milling machine, so you are ready to start machining your name-badge. As the way to do this depends on which milling machine you use, not all necessary information can be given here: please consult the manuals of your NC milling machine as well.

We have created toolpaths for a block sized 100 x 36.36 mm (4 x 1.45”). The block that we will actually use needs to exactly match that size, or be a bit smaller: if it is too large a ridge of material will remain on one or more sides, outside the white background area of the bitmap.
As we have not bothered about the block thickness (the Z) DeskProto has used the min and max Z-value (-1 and 0) and reports a block thickness of 1 mm.
In practice we will use a thicker block: otherwise only three separate characters would result.

For this job the cutting forces will be very small, so we can easily “clamp” the material block on the machine using doubly-sided adhesive tape. When using a clamp, machine vise or similar make sure that the cutter will not collide with the jaws of the vise.

Load the correct cutter in the machine’s spindle (the V-shaped cutter), and set the WorkPiece zero point with the tip of the cutter toughing the top of the material. For X=0 and Y=0 use the point exactly on the corner of the block when it’s size is exact, if not the set this point just outside the block. We used a white material with a blue top layer. In order to avoid scratching this blue to we have set the Z=0 about **0.2 mm above** the top of the material block.

Now you can open the NC file in your machine’s control software and start machining.

4. Bottle (geometry: two halves)
Lesson Four

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

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

Start a New Project

In DeskProto three types of project are available: Vector project, Geometry project and Bitmap project. The only difference between these types is which Operation is present: in a Vector project a Vector operation is present, etc.
For this lesson you need a Geometry project: create one using the command File > New Project > **New Geometry Project**. One of these three types is the default project (shown in the File menu in bold), simply pressing the New button on the toolbar will create a project of this default type.

Now use **Load Geometry File...** (File menu) or use the Load Geometry button to load the geometry file for the Bottle. It is located in the DeskProto Samples folder: just as in the previous lessons you can use the checkbox in the Start Screen to easily find that location.

Select sample file *Bottle.stl* (inch users choose *Bottle_inch.stl*). Note that Windows may hide the extension '.STL' and call the file a "*Certificate Trust List*". You can ignore that false information and just open the file.

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

Check the geometry

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

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

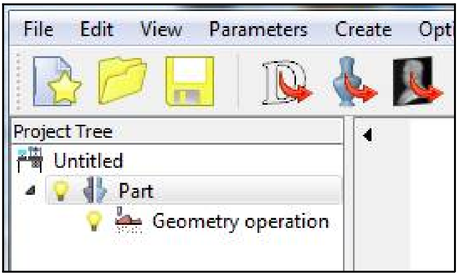
The **dimensions** can be checked using the button Part Information: the button on the DeskProto toolbar with the yellow [**i**], tab Geometry. For this file the dimensions should show a model of ca 54 x 29 x 86 mm: a nice perfume bottle, which (after rotating) will fit the working area of your milling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

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

Edit Project parameters

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

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



In the Parameters menu select Project Parameters. Much easier is to simply double-click the project line (first line) in the tree. The Edit Project Parameters dialog box that will pop up does not contain many parameters. On tab General it shows the ‘Filepath’, the name of the Machine and the names of all Parts.

The **Filepath** field for this project will be empty as the project file (*.dpj) has not yet been saved. In the Tree you can see that the new project does not yet have a name: it is called 'Untitled'. When you save the project for the first time you can enter a name, that will be used both for the file and for the project tree.

The **Machine** that is selected will be your machine, as when first starting DeskProto you have selected your machine as default. If not you can select a different machine for this project here. The default machine can be changed in the Default Project Parameters (Options menu).

A **Part** is machined in one fixation of the material block. In case you need more than one part you can add new parts here. For many projects (like in lesson one) one part is sufficient. For more complex models more than one part needs to be milled: like for an electric drill you will separately mill the right side and the left side, to be glued together later. For this bottle however both parts are equal, so in the software one part is sufficient.

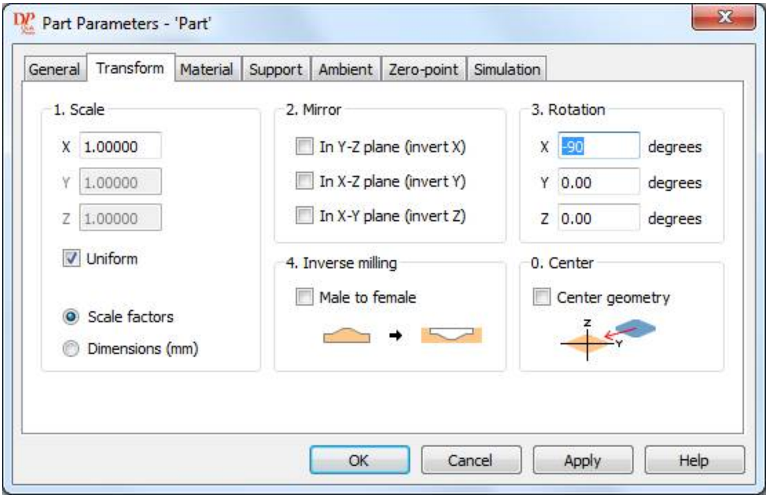
The other tab pages allow you to load and unload more CAD files, and to set Chaining settings. For now just ignore these other tab pages: you can leave the dialog box using the Cancel button

Edit Part Parameters.

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

The Part Parameters are presented using a number of Tab pages.
Part Parameters define the geometry to be milled.
Earlier in this lesson, when viewing the geometry, we already concluded that the geometry was not correctly orientated. You will do so now: the geometry rotation is one of the part parameters. In the Free edition and the Entry edition not all tab pages and not all options on each page are present.

The first Tab page (General) does not need any changes for this project. If you like you can change the name of the part, though that does not effect the toolpaths. The number of operations does not need to be changed: one operation is sufficient here.

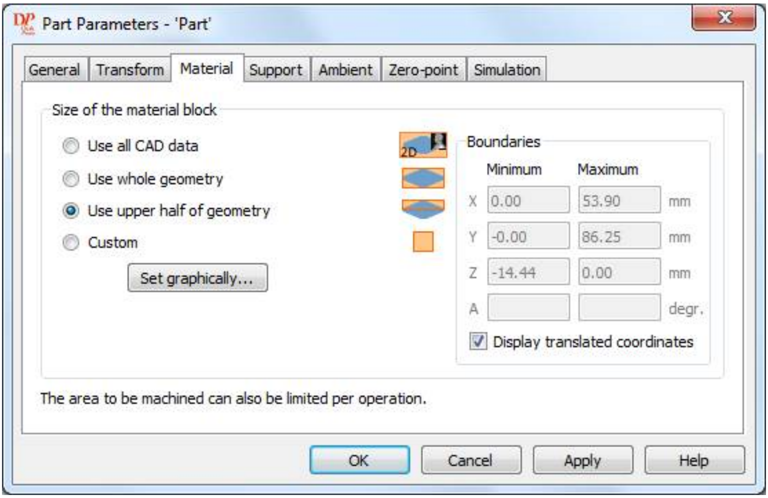


For this bottle model we need to set parameters both on the second tab (Transform) and on the third (Material). The **Transform Tab** shown above makes it possible to change the size and orientation of the geometry. For this bottle (as said) the orientation is not correct: enter a rotation of -90 degrees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

Using the Apply button is not needed: pressing OK also causes an implicit Apply. Still it may be handy in order to see what happens: if this is not what you need you can enter a different number without needing to reopen the dialog. On Apply both the drawing and the dimension shown in the Part Information dialog will be updated.

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

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



The third Part Tab is Material, and for this part the default (Use all CAD data) needs to be changed. The geometry of the complete bottle has been loaded. However, using a three axis milling machine this complete geometry cannot be milled in one part: the cutter cannot reach the bottom half. As stated before the model will be milled in two separate halves, so we now want to calculate toolpaths for half a bottle.

DeskProto uses a rectangular block of material, that can be defined by it’s minimum and maximum vales for X, Y and Z. By default the material block includes all CAD data: it is the exact *bounding box* around the data (vector, geometry and bitmap) that you have loaded. Making the material block smaller means excluding some of the CAD data. We want to machine half a bottle, which is easy as it is a predefined option: select option 3: “Use upper half of geometry”. You can see that when pressing Apply the Minimum Z will change to -14.44 (for inch users -0.57”), and that the orange block on the drawing will change as well.
Next press OK to exit this dialog.

Three notes on the Material block:

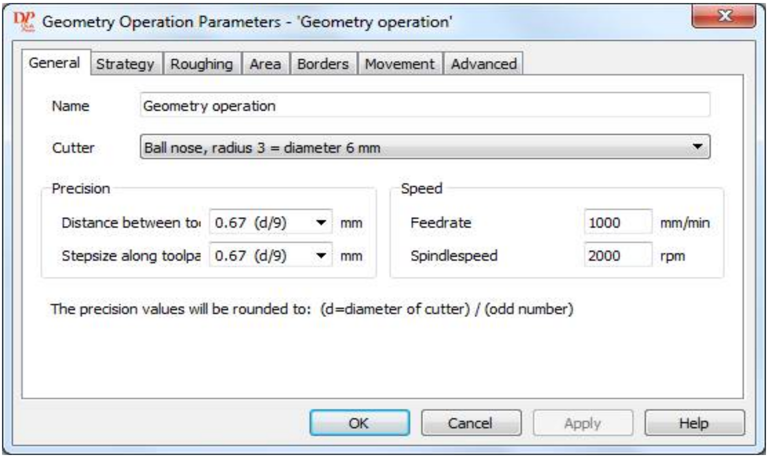
1. All values for the material block are shown in *transformed coordinates* (so after the changes as defined on the Transform tab), not in the original geometry coordinates. After these transformations one more change will be applied: the Translation, to be specified on tab Zero-point. By default DeskProto will show the *translated coordinates* here: the coordinates as used on the milling machine. You can instead use the *transformed coordinates* by un-checking the box “Display translated coordinates” (not present in the Entry Edition).

2. When you select option Custom for the material block you can define any rectangular block by entering X, Y and Z Boundary values. Easier still is to define the block using the mouse, which is possible via the button “Set graphically ...”.

3. You can of course also make the material block larger than the default bounding box, as the actual block that you will be using may be larger too.

Edit Operation Parameters.

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



For this dialog as well: in the Free editions and the Entry edition not all tab page nor all options on each page will be present.

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

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

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

The default Feedrate (traveling speed of the cutter) and Spindle speed (rotation speed of the tool in rpm) will in most cases be correct as well. In fact optimum values depend on the type of material you want to cut, however, when cutting light materials these values are 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

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

After calculating a dialog will pop up that gives a (rough) estimation of the machining time for your current settings. If not you can call it via "**Estimate Machining Time**" in the Create menu (in that case the "Always show" checkbox then was not checked). Note that the estimation is indeed rough: see the Help file for more information on why it is rough and on how you can calibrate it.

You will see (in case you used the cutter that was just suggested) that two different horizontal layers of cutter movements have been calculated: the first at level Z = -15 mm (0.4"), the second at the final depth. These are Roughing layers: the cutting length of this 4 mm ballnose cutter is 15 mm (for the 1/8" ballnose it is 0.4"), 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.

Important: in the **Free edition** this extra layer is not present, as this edition does not support Roughing. So take care: when your model is too high for the cutter, users of the Free edition need to run the NC file several times: start with the zero point defined (far) above the block, and then each next time with the zero point a bit lower until you have machined the complete part.

You will also see some horizontal dashed lines in gray, drawn over the geometry. These are positioning movements in Rapid mode, for instance from the end of layer 1 (maximum Y) to the start of layer 2 (minimum Y). When the optimization option **Sort** (on the Movement tab, not present in the Entry edition) has not been checked many more positioning movements will be present for the second layer.

Note: attentive readers will have noticed that the height of half a bottle is in fact only 14.44 mm, so less than 15. They also will have seen that the toolpaths go below the minimum Z-value of the material block. To clearly see this: draw a side view and compare the orange line of the block to the red lines of the toolpath. The explanation is that when using a ballnose cutter it needs to travel the Radius of the tool below the bottom of the part (so in this case 2 mm deeper). This is needed in case of (almost) vertical walls, which otherwise could not be machined completely. This is of course very important to remember when starting the milling machine, we will come back on it later.

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

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

Optionally add a Roughing Operation

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

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

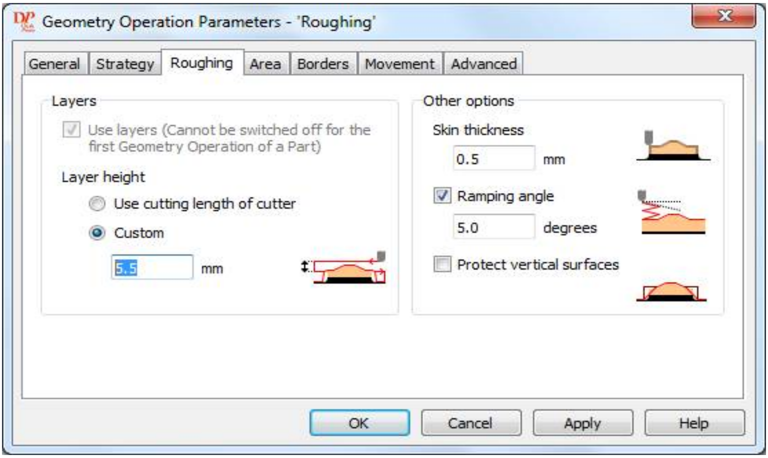
So first you need to Add an Operation. This can be done by right clicking the line "Part" in the Project Tree and then selecting Add Geometry Operation in the context menu. Or as an alternative you can Add or Copy Operations in the

Part parameters dialog.

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

Now the sequence of the operations is wrong: roughing of course needs to be done before finishing. You can fix this in the Part parameters: using the black arrow buttons called "Move" on Tab page General you can change the sequence of the operations. In order to do this you have to select an operation (make its line blue) first. Alternative is to use the context menu in the Project Tree for one of the Operation lines, which will offer the options Move up and Move down.

Note that after this change the toolpaths for Operation Finishing will still show the layers (these were switched on automatically because it was the first operation, however now it is the second one). Open the Operation parameters for this operation (double-click it's line in the Tree), on tab page Roughing uncheck the option "Use layers", close the dialog with OK. The toolpaths then will be recalculated, now without layers.



Next you can set the parameters for the Roughing operation. Open the Operation Parameters dialog for this operation. On tab General you need to select the correct cutter: as said before you can use the same 4 mm diameter ballnose cutter (or 1/8") as used for finishing. Then you can add the actual roughing parameters, on tab page Roughing:

- The **Layer height** thickness can be fine-tuned: select Custom and enter a value. For light materials like foam or tooling board you can enter 5.5 mm (0.22”) to have three layers that are about equal (16.44 / 3.0). For stronger materials like perspex or metal you will need to use a smaller value.

- You can set the **Skin** (to be removed during finishing) at 0.5 mm (0.02”). The skin is an allowance of extra material that remains present around the model. Any damages that may result of the roughing toolpath will be in this skin, to be removed when finishing.

- **Ramping** concerns how the cutter travels to the first point that needs to be machined. Standard the cutter first travels to the correct XY and then just plunges into the material, moving along Z. When ramping this drilling movement is replaced by a declining movement (see the illustration in the dialog), for which you can set the gradient angle. This will make cutting much easier than in a pure drilling movement, especially when machining metal.

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

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

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

1. For Roughing it is more efficient to choose a different **Strategy** (second tab page of the operation parameters): when you select strategy Block instead of Parallel the toolpaths will proceed form the outside of the block to it’s center..

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

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

At this point you may want to draw a Simulation, in order to check if the (simulated) resulting part is indeed what you expect. When you first calculate for Roughing and then for Finishing you can see the result after both steps in the process.

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

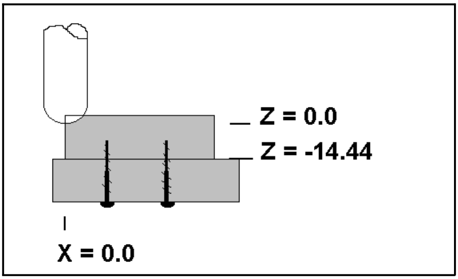
To the milling machine

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

The plan is to (twice) machine exactly half a bottle, the flat bottom surface of the block being the plane of symmetry, to be used to join both halves. Fixturing faces the same problems as described in Lesson 2: the cutter will cut on all sides of the model, and the tip of the (ballnose) tool will come **below the bottom of the block** and may damage your machine’s working table. The solution is the same as in Lesson 2: use a slab of wasteboard below the block, and connect both using a few screws from below. See the illustration below.
Note: in case of a light type of material like PUR-foam screws are not needed: use double-sided adhesive tape to attach both blocks to one another and to fix them on the machine table.

Make sure that:
- the lower block (the wasteboard) has its top plane and bottom plane exactly parallel.

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



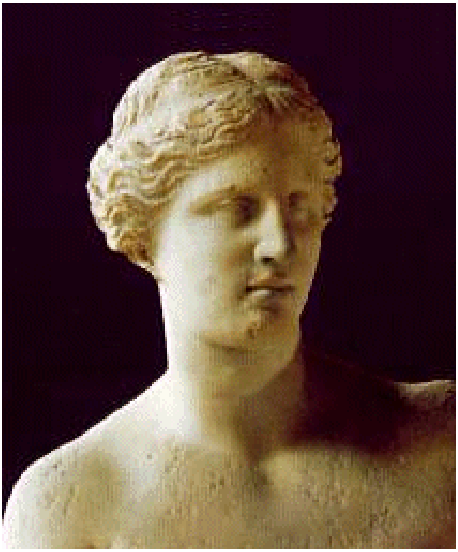
Now you can fix the blocks on the machine, and enter the (0,0,0) position. For X and Y this is done just as in Lesson 2, for the Z this is different. While in the previous lessons you entered Z=0 on the top of the block, now you start with the bottom of the block (which is the symmetry plane of the bottle). Position the tip of the tool on the same height as the bottom of the block. From there go up 14.44 mm (for inch users: 0.57”) and set the Z=0 level there. You can read this value 14.44 from in the DeskProto Part Information dialog box: Tab 'Material' shows that the minimum Z of the prototype is -14.44 (the symmetry plane) and that the maximum Z is 0.

The Z-level of the actual top of the block is not important now: it is OK as long as it is not below Z=0. Obviously it must not be too high either, as the cutter must be able to remove the extra material above the model top.
In case your block really is too high you can deal with that in DeskProto, by setting the maximum Z dimension of the Material block for this part to a higher value.
Remember that when using the Free edition it may be needed to start with a zero point above the top of the block (as no roughing layers are present).

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

5. Venus (geometry: rotary axis)

Lesson Five



In the fifth lesson you will learn how to create toolpaths for a rotation axis: an extra device on your CNC milling machine that lets the object rotate during machining. It is also called Fourth axis or A-axis, and it looks like the spit on your barbecue. This lesson of course only is useful in case you have a machine with such 4th axis. Rotation axis machining is available only in the DeskProto Multi-Axis edition.

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

Start a New Project

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

The geometry file that you have to load is called **Venus.stl**. This file is not present in the Samples folder, you will first have to download it from www.deskproto.com The file is only available in mm, so inch users will have to scale down with factor 0.03937 (Part parameters, tab Transform) – though in fact a smaller factor will be needed in order to create a scale model.
Load the file via Load Geometry file (button or File menu command).

The rotation axis on most machines is parallel to the X direction, and is then officially called the A-axis. You can tell DeskProto that your machine has a rotation axis by editing the machine definition in the Library of machines (Options menu): in the Advanced settings check “Has rotation axis”. On one side of the axis the rotary table’s vise or chuck holds the workpiece, often on the other side a tail-stock is available to support the far end.

DeskProto also supports machines with the rotation axis parallel to Y. For such machine you can check the option “Swap X and Y coordinates” in the Advanced machine settings mentioned in the previous paragraph. You will then see a second Orientator on your DeskProto screen, showing the orientation of the coordinates on your ‘swapped’ machine. All user input needs to be done in the DeskProto orientation, the NC output will be in your machine’s orientation. This sounds complicated, however you will get used to it real soon.

When you view the Venus geometry after loading, first thing that you will observe is that the geometry is not orientated correctly. The most logical rotation axis for a head is the line from neck to top (so a vertical line when standing upright). This line now is along the Y-axis instead of along X as needed, so you will have to **rotate the geometry minus 90 degrees around Z**.

When you check the part’s dimensions (Part Information dialog) you will see that the part is very large: the unscaled geometry is 284 x 421 x 315 mm. So it probably won’t fit inside the working area of your machine (DeskProto will pop up an error). You will have to **scale down**, the scale factor depending the part size your prefer – and of course on your machine’s dimensions.

So open the Part parameters dialog, and on page Transform set the correct Rotation and the correct Scale. Instead of scaling factors you can also enter the desired dimensions here. Note that the dimensions are shown as they are

before rotation: the Transform options are applied in the sequence as indicated by the numbers (0, 1, 2, 3, 4). So independent of the rotation settings that you entered the highest dimension of this Venus geometry will be for Y, just as in the STL file.

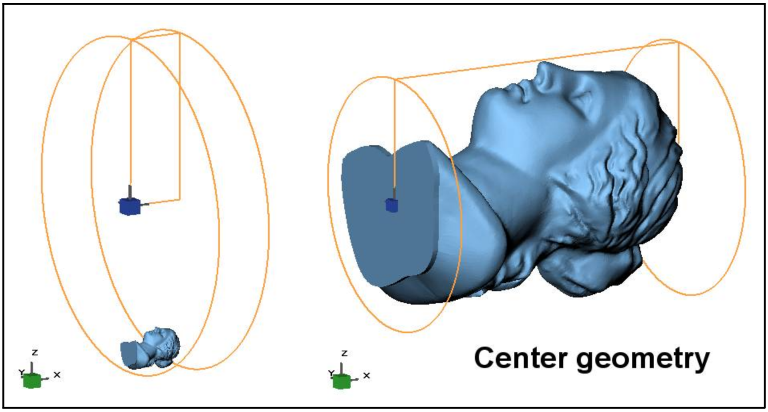
In this lesson you will use **Continuous Rotation** axis machining, where the part rotates during machining. DeskProto will then produce XZA toolpaths: only the X, A and Z axes are used. The Y-axis does not move: the cutter needs to be positioned exactly above the rotation axis when starting to machine (Y=0.0), and remains there all the time.
The alternative use of a rotation axis is **Indexed machining**. Here plain XYZ toolpaths are applied, from different sides, with an A-rotation for positioning in-between. DeskProto supports indexed machining in the wizard called *One or more sides, automatic rotation* (The “N-sided wizard”). That wizard will automatically generate a number of parts: one part for each side to be machined.

Use the Rotation axis

The next thing that needs to be done is to check the option **Use Rotation axis (XZA paths)**. This option can be found on the first Tab of the Part Parameters dialog box. Of course this option is active only in case you selected a machine with such optional 4^{th} axis. This means that in the DeskProto machine definition it must have been configured (obviously your actual machine must have one too). So if this option is grayed out then go to the machine library (Options menu) to fix this in the machine definition. In case this option is not present at all then you are probably not using the Multi-Axis edition of DeskProto.

Do not confuse a rotary axis with a lathe: on a CNC machine with rotation axis the cutter rotation provides the cutting movement, the part rotates only for positioning purposes. On a lathe the cutter does not move: the cutting movement is provided by the part that rotates at high speed.

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



After switching on the Rotation-Axis and pressing OK (and acknowledging an error, see below) you will see that a completely different material block is drawn on screen (in orange lines). No longer the rectangular block that you are used to, but instead a cylindrical block. This does of course make sense for rotation axis machining.
DeskProto does not support a rectangular block for rotary machining; when you are using such block you need to define a cylinder block large enough to include your rectangular one.

Despite the scaling down that you just applied, still a **size error** will be given. The actual part is small enough (because of your scaling), however the material block that DeskProto calculated is far too large. See the illustration above. The geometry is rotated round the “real” X-axis (so the axis at position Y=0, Z=0). For the Venus STL file this rotation axis is located outside the geometry, so a very large cylinder block will result and cause this second size error.
In DeskProto an extra option is present to solve this: on the Transform Tab of the Part Parameters check the option **Center geometry**. This will result in the rotation axis being placed though the centerpoint of the part. Now DeskProto will accept the part settings without errors, and the material block drawn on screen will be the one that you need.

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

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

As said in the message-box just mentioned DeskProto has already set the Z Translation to None: Z=0 is with the tip of the cutter on the rotation axis (the convention most used). The result can be seen in the Part Information dialog (Tab page Geometry), and also (easier) in the position of the **blue cube** (the "Orientator on the WorkPiece zero point") on your screen. Make sure to set this WP zero point on your machine accordingly: an incorrect setting may damage part, cutter and/or machine !!

Two more parameters have been changed automatically:
The **Material** block (Part parameters) has been set to Use upper half of geometry. When rotating the geometry the actual content of this "upper half" will change all the time, so this setting will cover the complete part (at least for most geometries).
The **Borders** (Operation parameters) have been set to 'No extra', for all operations. You can find this parameter on the Borders tab of the Operation parameters. Its normal value is 'Extra for cutter', as normally all outside surfaces of the model need to be machined. For rotation axis machining the model needs to remain connected to the rotation axis unit and to the tail-stock. So the outside surfaces left and right may not be machined, which is achieved by this setting.
For some geometries it will be handy to add **Support tabs** (Part parameters): extra material left and right, for a better connection to rotation axis unit on the left and to the tail-stock on the right. As the Venus geometry has one flat side (now on the left) such blocks are not needed for this project.

When the rotation axis is used, for some parameters in DeskProto the Y-axis setting (to be set in mm) will be replaced by an A-axis setting (to be set in degrees). Note that DeskProto replaces Y by A, so in fact what you do remains 3-axis machining. You can choose to use either XYZ (normal) or XAZ (rotary); DeskProto will not use all 4 axes simultaneously. Nevertheless 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 coordinate values of the Material block. For XYZ machining these values simply define the outer limits of the 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 doughnut shaped block, a Min Z below 0.0 will result in toolpaths below the rotation axis.

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

Fine-tune the parameters

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

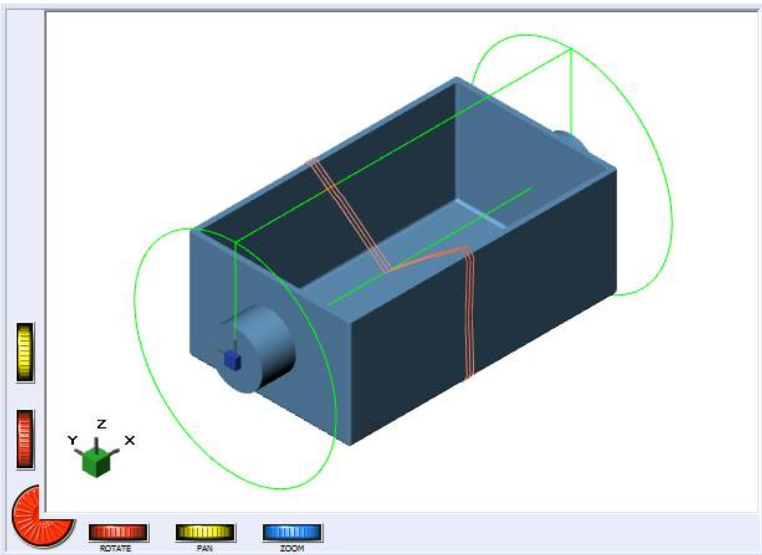
As said before, all parameters concerning the A-axis are in **degrees** instead of in mm. This does not apply to the precision settings though: for toolpaths in the X-direction the Distance between toolpaths should be in degrees, for toolpaths in the Y (A)-direction the Stepsize should be in degrees.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

With 3-axis XYZ machining the tool cannot mill the bottom side of the part, as 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 would both be damaged and unfinished. DeskProto will take care that this does not happen, by assuring that some connection will remain present between the rotary table and the model, or rather by popping up an error dialog if such connection is not present.

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

Without such flat surface you may want to define two support tabs (when rotation axis machining has been selected the default supports will be two cylinders). Note that then the Borders need to be switched off too, in order to prevent machining the flat outside surfaces of the support tabs.

For this Venus lesson :
- switch off the Borders to keep the left side connected to the rotary chuck.
- do not use support tabs
- do not use the tailstock: only the left side is stable enough for this small part.
- make the Material block’s Max X a bit larger to allow the cutter to completely machine the head even with the Borders switched of (in fact you then manually add a border on that side).



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

The screenshot also illustrates the presence of **undercuts** for rotation axis machining. Undercuts are areas where the cutter cannot reach. For XYZ machining this is the bottom side of an object. For rotation axis machining this is different: see the three toolpath lines (in red) drawn in the picture above. Only the X, Z and A axis will move: the Y-position is constant, with the cutter positioned exactly above the rotation axis. This means that the cutter cannot completely empty the box: it just cannot reach two of the vertical walls on the inside of the box. The V-shaped toolpaths that are drawn illustrate this. For this geometry it will be better to machine from two or from four sides, also using the rotation axis: indexed machining.

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

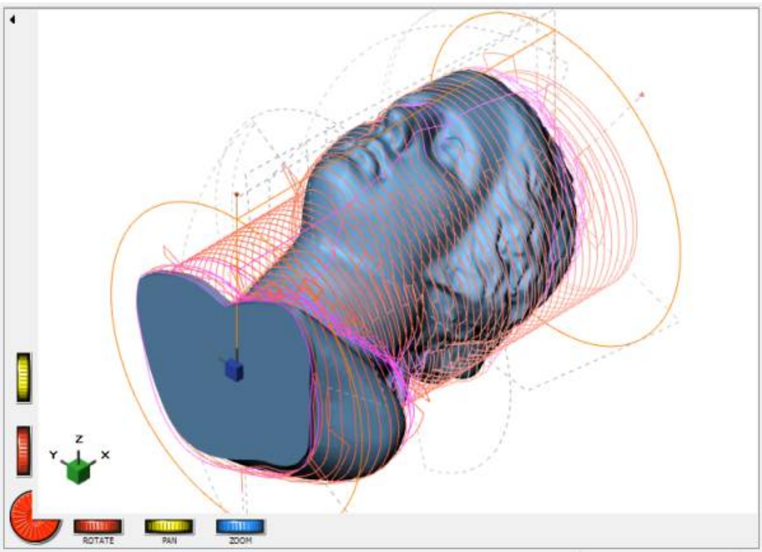
Finally: perhaps you want to use toolpaths in the A-direction, combined with either conventional or climb machining. This way the rotation axis can remain rotating in the same direction (for instance from 0 degr. to 360, next 360 to 720, etc). This is possible only when in the machine definition (Options menu) the option "**A-values may exceed 360**" has been checked. 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.

Keep in mind that some rotation axis units can only keep rotating in one direction for a limited number of times, meaning that they do not support A values that exceed 360. When this option is not checked after each toolpath the cutter will move up to Zfree safe height and travel back to 0 degrees to start the next toolpath. In fact the same as what happens with XYZ machining.

A great extra feature (new in DeskProto V7) is the **Helix** option: a checkbox for strategy Parallel (Operation parameters, tab Strategy). This checkbox is active only for toolpaths around the A-axis and when continuous rotation is in one direction (as described above) has been configured. Without this option checked the toolpath will complete a full 360 degree rotation with a constant X (only A and Z vary), move X to the next toolpath, complete the next toolpath at constant X, etc. With the Helix option checked both the X and the A coordinate keep progressing all the time, so the complete part will be machined with one continuous movement (X, A and Z varying). This will be faster, produce a smoother movement and ignore any backlash problems for X and/or A.

When combining continuous rotation with roughing layers a disadvantage is that Sorting becomes quite ineffective, as most sorting optimizations require a meander toolpath direction.

This was a lot of background information on rotation axis machining (thanks for your attention!), most of which is in fact not needed to complete this Venus model but may prove to be useful later.



The resulting toolpaths will look like the paths in the picture above, depending on your choices for scaling, cutter, precision, etc.
The picture shows a Roughing operation. You can clearly recognize a series of circle toolpaths forming a cylinder: the first layer. The gray dashed lines are the

positioning moves in the lower layers, skipping any area that has already been machined (when Sorting is off or when using a Helix you will see many more of these than in the illustration above). The bottom surface (left side) of the statue remains un-machined: the connection with the rotation axis. The toolpaths shown are in A direction and meandering, as you can see on the forehead.

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

For rotation axis machining DeskProto does not offer the option to simulate the result: the Simulation button will be disabled (grayed out). Such simulation requires a much more complicated calculation than for three-axis machining, and such algorithm has not yet been implemented.

To the milling machine

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

First you have to set the **WorkPiece zero point**. With the default translation settings this will be on the left side of the cylinder block (X=0), with the tip of the cutter exactly on the rotation axis (Y=0, Z=0). A=0 is not important, however it may be when your are using a rectangular block
Setting X=0 and A=0 is easy: move to the correct position and tell the machine to use that as zero. Setting Y and Z is more difficult as it is not easy to see if the tip is exactly on the correct location. Some machines offer a calibration help, like a horizontal surface at the correct Z-value. If not you can try to visually find the zero positions as accurately as you can. That will be OK for this Venus statue, however insufficiently accurate when for your part the cutter needs to travel below Z=0.

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

'touching' Y-values. They should be identical but for the minus sign. If not, then correct your Y=0 accordingly.

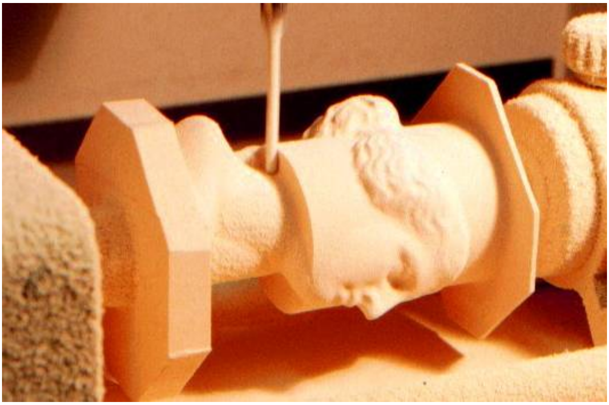
This is a lot of work: the bright side is that you need to do this only once: the same zero point will be valid for a next job as well.
Note that with this zero position you must take care not to let the cutter move to the workpiece zero point (which is a standard command on many machines), this will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

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

Next you will have to **fixture** your material block, using the available options of your rotation axis, like a 3-jaw chuck, pins on a circular plate or a drill-head. Use the tail-stock with a centering pin on the opposite side if one is available, as this makes the stability of the model much better. Not for this Venus model though: we have set the toolpaths to completely machine the top of the head. For machining rings (in wax, for investment casting) you have to use some special fixturing tool, or use a hollow wax bar that can be fixtured in a 3-jaw chuck.

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

Many controllers have problems in setting the correct Feedrate when rotation axis machining, as the rotation speed that is needed to achieve a certain linear speed depends on the distance between the cutter and the rotation axis. DeskProto offers (this is new in Version 7) two special options for the rotary feedrate: in the postprocessor you can configure DeskProto to use “Inverse time Feedrate” or “Angular Feedrate” for rotary movements. For more information about this see the Help file and the FAQ on the DeskProto website.



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

The machining process will look like the photo above. Note the cylinder form created by the first layer. After machining you can remove the remaining block on both sides using a simple saw or band-saw

A picture of the result :

6. Cellphone (geom: two-sided)

Lesson Six

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

The example used is the front panel of an old 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.

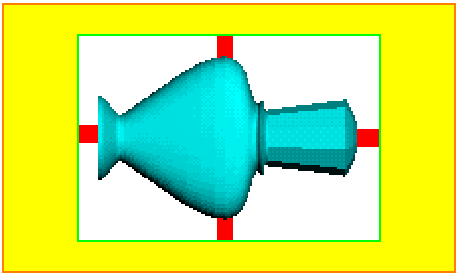
The cellphone STL is a large file (8 Mb), so it is not included in the standard DeskProto Samples. You can download the file from www.deskproto.com

Two-sided machining

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

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

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



Generally speaking, when milling a model from two sides the problem is the second side: how to fixture the model, and how to do this in the correct orientation for that side, at a position that matches the WorkPiece Zero point. The DeskProto Wizard solves this by using an over-sized material block, so large that a complete frame remains present around the part. Four small bridges are added to keep the part connected to this frame, called **Support tabs**. These can be manually removed later. See the illustration above. This frame makes 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.

The two-sided Wizard

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

Two more options are offered on this wizard page: Scaling and Rotating. For metric users **Scaling** is not needed as well (unless of course you want to make a scale model). As said inch users have to apply a scaling factor of 0.03937 (uniform scaling). For the Cellphone the **Orientation** may remain as default, with the top surface on top: rotating the geometry is not needed. Note the light yellow question mark icon, left of each question: position the cursor on this icon to get help on the question. This Wizard tooltip also tells you where that same setting can be found in the dialog based interface (so without the wizard).

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

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

The **Milling depth** is in fact the setting "Minimum Z" for the Area to be machined (Operation parameters). The same depth will be used for all operations. Default depth is halfway down the block: for the cellphone geometry a higher depth value is needed in order to completely machine the geometry. Also: do not forget to change the depth after changing the block thickness, as the depth is relative to the top of the block. Dimensions and depth will be drawn on screen as line drawings: orange lines for the block, green lines for the area to be machined.

We used a slab of tooling board of 25 mm thick, and set the milling depth to 17 in order to completely machine the part from both sides. You can change the drawing to a side-view and then in the wizard press Apply to check if the milling depth setting is correct.

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

Which **Cutter** to choose is an optimization between many factors:
- cutting speed (a large cutter can remove material more quickly)
- geometry characteristics (for freeform surfaces use a ballnose cutter, for horizontal and vertical surfaces use a flat tipped cutter).
- surface quality (a larger ballnose cutter will create a smoother surface)
- small details (for small inner radii a small cutter is needed)
- height of the model (small cutters are short)
- use the same cutter for all operations or accept a tool change.

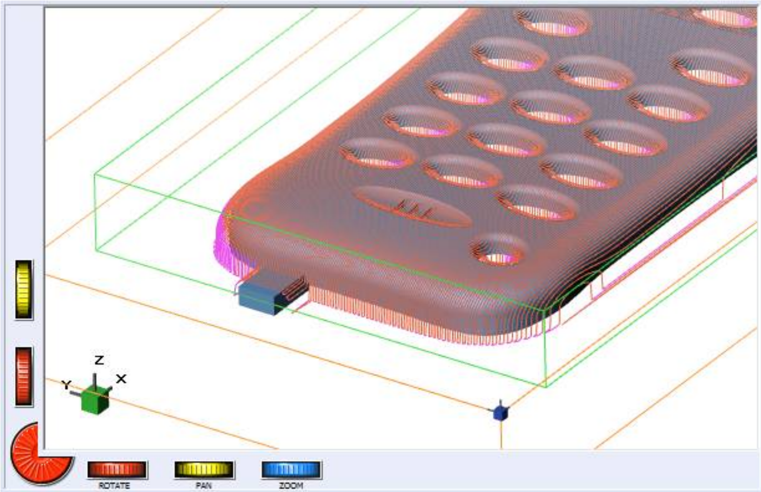
For this cellphone part we have used a 3 mm ballnose cutter (radius 1.5 mm), for all operations, in order to avoid a toolchange. 3 mm diameter is sufficient for most details here, and the cutter is long enough to machine the complete model. For inch users a 1/8 inch diameter ballnose will be a good choice.

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

The default **Block** strategy is optimal for most geometries, as it will minimize the number of positioning moves during the lower layers.
The default **Skin thickness** is 10% of the cutter diameter, and the default **Layer height** is 100% of the cutter diameter. In most case these values will be OK as well.

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

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



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

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

In this wizard *two* buttons are available to Save the NC program file: one for each part. Choose file-names that indicate which side of the part it is for.
The buttons to directly send the toolpath to the machine are available only if that option has been configured (in the Preferences). These can of course only be used when your machine supports this option. The Send to machine option is not present in the versions for MacOS and for Linux.

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

Fine-tune the parameters

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

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

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

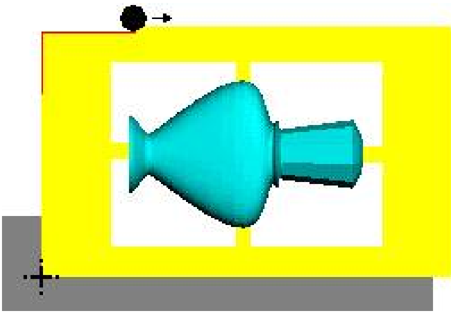
In the Part parameters you must not change the size of the material block, as this size is needed to exactly match both sides and set the WorkPiece zero point. This WP zero point will be the same for all operations in the part. The support tabs may be changed. Obviously the support tabs for both sides must exactly match, and to avoid any differences DeskProto has selected the option “Use tabs of 1st part” in the second part.

The Simulation that DeskProto offers for a two-sided project is a separate simulation for each side. A combined simulation, showing the result when both sides have been machined, is not (yet) possible.

To the milling machine

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

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



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

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

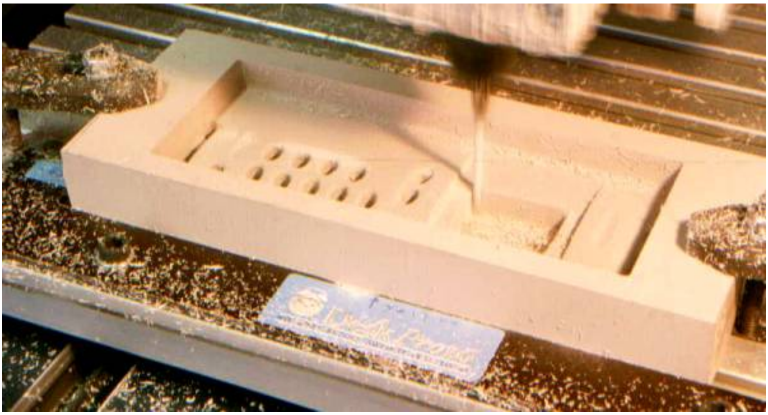
You can machine both reference planes using manual control (mind the radius of the cutter when doing so!). DeskProto does not machine these reference planes automatically, as it does not know where you have placed your clamps.

For instance for a cutter of 6 mm diameter: move the cutter to X=-3 and a suitable Z, and then move the Y to machine a flat plane on the left side of the block. Next move the cutter to the prescribed Y-position + 3 mm (cutter radius) and move the X to machine a reference plane along the complete backside of the block.

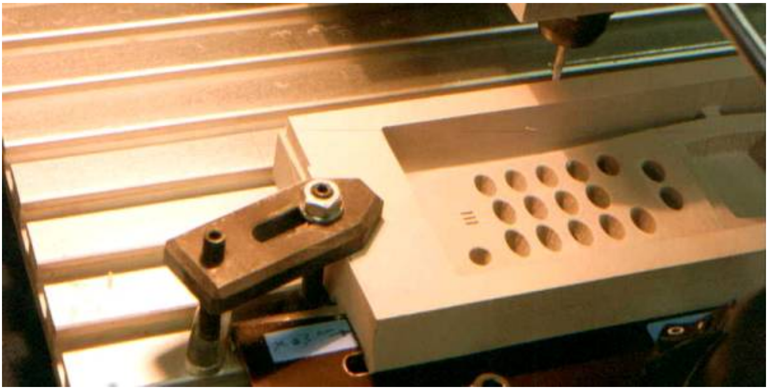
After machining these reference planes you can loosen the clamps and take the block off, turn it upside down ("flip" the block), press these two reference planes against ruler and end-stop, and finally again fasten the clamps. This makes sure that the block is exactly lined up with the machine, and is exactly at the correct position.

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

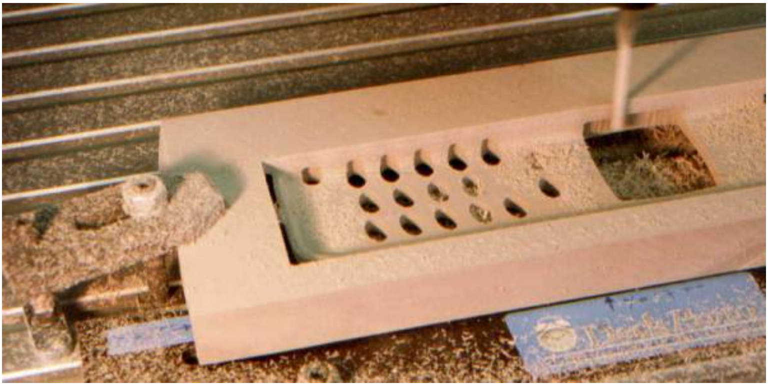
Below you can see a few illustrations that show this process for the Cellphone model.



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



Before loosening the clamps, first you have to machine two reference planes on the left and on the back, as shown here. This is why some oversize is needed on the back side of the block. Also the left side of the block needs to be a bit inclined to have some excess material (body) 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. Note the two machined reference planes and the support tabs (bridges): both clearly visible. The frame and tabs have to be removed manually: some sanding will be needed where the support tabs were attached to the model. The rest of the model will already be smooth enough.

7. DP bottle (vector: advanced)

Lesson Seven

The lessons so far have all been about Vector data **or** about Geometry data **or** about Bitmap data. A very nice feature of DeskProto is that you can combine two or three data types in the same project. For instance use vector toolpaths to machine some detail in a geometry project.

DeskProto even allows you to let two data types interact: both vector toolpaths and bitmap reliefs can be projected on a 3D geometry. In this lesson the 2D DeskProto logo will be projected on the 3D bottle geometry that was used in lesson 4.

The lesson starts with some background information on vector data and toolpath types for vector data.

Vector data files

For vector data 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 export EPS (postscript). Of both file types only a subset is supported:
The **DXF** subset includes point, line, polyline, LW polyline, arc, circle, ellipse and spline.
The **EPS** subset includes point, lineto, curveto and moveto.

DeskProto will convert arcs and splines to polylines, **arc** movements in the NC code (G2 and G3) are not (yet) supported.

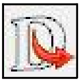
Vector files in most cases are 2D files, so will not contain any Z coordinates. DeskProto will draw the 2D curves **at the top of the block**, also when for that part you have set the Z=0 at a different height (this is different from previous DeskProto versions). The machining depth that you enter for the toolpaths is calculated from the top of the block.

Nevertheless it is also possible to open a DXF file containing 3D vector data. In that case DeskProto will ask you whether or not you want to use these vector Z-values. If yes, the Z=0 in the vector file will be aligned with the top of the block, and the machining depth that you enter for the toolpath will be calculated relative to the Z-values in the vector curve. The result will be a 3D toolpath. So in order to identically position Geometry data and 3D vector data you need to place the Zero point at the top of the block.
The decision whether or not to use the vector data Z-values can also be made later: in the project parameters, tab Vector.

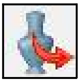
DeskProto offers three toolpath types: Profiling, Pocketing and Drilling. The first two have been covered in Lesson 1, drilling has not yet been mentioned. In the Vector operation parameters you can select curves for each toolpath type:
Profiling will accept all curves in the vector data, except points.
Pocketing will only accept curves that are closed (contours).
Drilling will accept: points (centerpoint of the hole to be drilled), “+” signs (two lines that each have the cutter diameter as length, intersecting at the centerpoint) and circles that have the same diameter as the selected cutter.

Combining 2D vector data with 3D geometry data

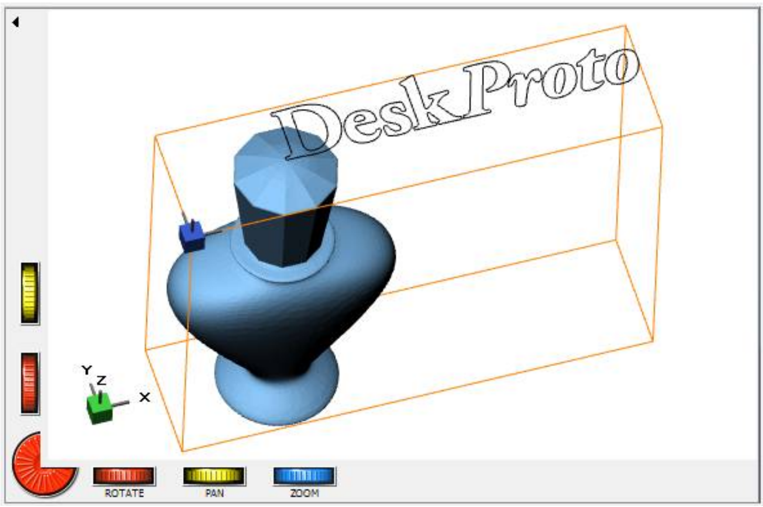
In the previous lessons you have seen that the Part parameters are used to define what will be machined: scale, orientation, etc. The available parameters to achieve this are a bit different for each of the three data types. For instance rotating round X and Y is not supported for vector data, so the Transform tab offers different options for vector en for geometry. What to do when both data types are present ?
DeskProto has solved this by showing two sets of settings: Vector settings and Geometry settings, which will be shown in this lesson (if bitmap data is present the Bitmap settings are the third set that is present).



For this project we need two types of CAD data: Vector data for the DeskProto logo, and a Geometry for the perfume bottle.
First load the Vector file: sample file *2D_DeskProtoLogo.dxf*

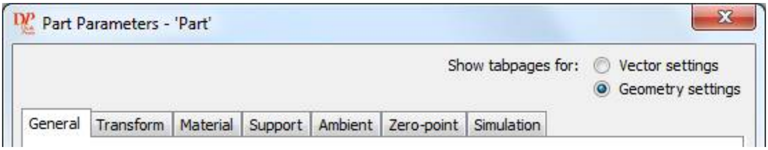


Next load the Geometry file: the same file *Bottle.stl* that we have also used in Lesson 4.
You will see that the block of material (the orange lines) will be exactly large enough to include all CAD data, and that the vector curves are displayed at the top surface of the block.



The X and Y position of both datasets matches their positions in the original CAD file. For the bottle the zero point in CAD (and also in STL) is located on its centerline, and for the logo it is located on it’s base-line just left of the letter D. The Z=0 in CAD is located below the bottle, and the logo is drawn at the top of the block. In DeskProto you can display this zero point by checking “CAD zero point” in the Items visible dialog (though in the above view it will be hidden below the bottle geometry).

The WorkPiece zero point in DeskProto (the blue "orientator") is not the same as the CAD zero point because in the Part parameters (tab Zero-point) an automatic translation is made to move the CAD data to "Positive X&Y for material block" and "Top of block zero".



When you now open the Part Parameters dialog you will see that above the row of tabs a new "Radio button" option has been added: you now can choose to show the tabpages either for the Vector settings or for the Geometry settings. Some of the tabs are equal for both settings, like tab General and tab Zero-point), other tabs are different (like Transform). Also the Geometry settings include more tab pages than the Vector settings.
The transform options can be used to position both datasets relative to one another, as will be shown in the next paragraph.

A nice detail is that vector machining can also be used with rotation axis machining: the 2D vector drawing then will be wrapped around the 3D cylindrical block of material, like a label around a jar of marmalade. That way it is easy to engrave for instance a name on the round sockle of a bust that you have machined.

Projecting 2D contours on 3D geometry

As you have seen in Lesson 1, DeskProto's vector toolpaths can be used to engrave 2D text on a flat surface, this lesson will show you how to engrave on a curved surface. This will be done using a nice feature of a DeskProto Vector operation: the option "Project vector curves on 3D Part geometry". When you check this option the resulting Machining level (Z-value) will not be relative to the top of the block (so at a constant Z-level), but instead it will be applied relative to the Z of the part-geometry at that location. The result will be that the vector curves are indeed projected on the 3D geometry.

You have just loaded Vector file *2D_DeskProtoLogo.dxf* (the DeskProto logo) and the Geometry file *Bottle.stl* (the perfume bottle). In this paragraph you will learn how to engrave this logo on the bottle's front surface.

Open the part parameters and select the **Geometry settings**. On tab Transform enter a Rotation around X of -90 degrees, and on tab material select "Use

upper half of geometry”. After each change you can use the Apply button to see what happens.
Next select the **Vector settings**. The DeskProto logo is too large for the bottle (even too large for the current block of material), so it needs to be scaled down. On tab Transform enter a Scale factor of 0.5 (Uniform checked, so both for X and for Y). The next step is to correctly position the logo over the bottle. This is called “panning” the vector data over the geometry data, and a nice Align option is present to make this easier. Select a Top view (the first of the 8 cube buttons on the toolbar) to properly see the position of the logo. Now on the same tab Transform, under Panning, use the button “Align to...”. In the Align Vector Data dialog select “Center” both for X and Y, select Geometry data to align with and press OK. In the Part Parameters press either Apply or OK to see the new position of the logo.

The DeskProto logo is nicely centered over the geometry. If needed you can now fine-tune. Perhaps a bit smaller is better: you can change the scale (after that you will need to again align). And the position could perhaps best be a tiny bit to the right (to visually compensate for the large D) and a bit lower on the bottle (to engrave only on the front surface, not on the shoulder). It is easy to play around by changing the two Panning values and then pressing Apply. When Scale and position are OK press OK to close the Part parameters.

To machine this part you will need one or more Geometry operations and one Vector operation. How to configure the Geometry operations (roughing and finishing, or finishing only if you prefer that) has been covered in the previous lessons, please go there if you need help in doing so.
Adding Operations can be done by right-clicking on the line Part in the Tree, and then selecting “Add Vector/Geometry/Bitmap operation”. Add operations to create a Part with one (or two) Geometry Operations and one Vector Operation, and then enter the Geometry operation parameters that you prefer.

Now open the Vector Operation parameters dialog. All geometry inside the logo’s contour lines needs to be removed, which is called Pocketing, so on tab “Pocketing” select All for select curves, and Offset as strategy. It will be clear that you need a small cutter to fit inside the curves, so select a 1 mm diameter cutter (ballnose or flat tip). Press OK to close the Vector Operation parameters dialog, make the Geometry operation(s) invisible by clicking on the yellow lamp buttons in the tree, and press button Calculate toolpaths (in case asked: Yes, only the visible operations).

You will now see the logo will not be completely machined: no (red) toolpath lines are present in the smaller logo sections, as even this small cutter is too thick to fit in the pocket there. You can of course select a smaller cutter, however you will soon see that for this project a 0.5 mm diameter cutter (or smaller) is needed: very, very thin !
Best solution is to use a conical engraving cutter. Also called V-cutter, as the

tip looks like a V. This V must have a flat tip, as otherwise pocketing is not possible (for pocketing the tip diameters is used to calculate the distance between the toolpaths). So select for instance cutter “Conic engraving tool 30 degrees” and try again. Now the logo will be completely filled with toolpaths, very close to one another. You can set a larger Stepover in the vector operation parameters, tab Pocketing, under Strategy detail settings. Default Stepover is 50% (of the tip diameter), changing that to 80% will reduce the number of toolpaths.

In a top-view the toolpaths that you now have may look OK, however when seen from a different viewpoint it is clear that they are not: all toolpaths still are at a constant Z-level. Again open the Vector Operation parameters, now open tab page Z-settings. Here as machining depth set -0.3 mm (-0.01”), check “**Project vector curves on 3D Part geometry**” and as Calculation precision set 0.09 mm (ca 0.004”).
Now the toolpaths should be OK to engrave the logo.

On your screen they will be invisible though, as they are below the surface of the bottle geometry. Switching off “Geometry, rendered” in the Items visible dialog will make the toolpaths visible. You can now save the NC file(s) and start cutting.

The image above has been tricked: we have set the machining level to +0.1 mm, in order to make a picture with visible toolpaths (these paths are of course not useful for machining).

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

On these tilted surfaces the logo may also be distorted by the cutter's thickness. DeskProto will project the 3D shape of the cutter on the 3D geometry, and when projecting on tilted surfaces the outside of the cutter will touch the geometry before the center of the cutter does. The cutting depth is set from this first contact, so not for the center of the cutter.

8. Lithophane (bitmap: advanced)

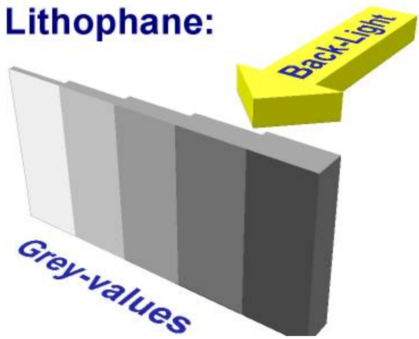
Lesson Eight

Creating lithophanes is a great application of DeskProto's bitmap machining feature. In this lesson we will explain what a lithophane is, and how you can easily create one using DeskProto. We used a famous portrait by Dutch painter Johannes Vermeer: **Girl with a Pearl Earring** (a public domain work of art, downloaded from commons.wikimedia.org).
DeskProto converts the 2D bitmap information in this photo to a 3D relief and then calculates toolpaths over this geometry.

The second subject of this advanced bitmap lesson is about combining a bitmap relief with a 3D geometry. Again the sample perfume bottle will be used, and in this lesson the relief of a shell will be added to the bottle's front surface.

Lithophanes

A lithophane is a work of art created in a thin sheet of translucent material (a sort of “milk glass”). The front-side of the lithophane is shaped in a relief, and when illuminated from the back side a clear image in black&white will become visible.



The above image explains how this works: the thinner the material, the more light can come through. The material can be a sheet of (for instance) 3 mm thick white plastic (PE, PVC, Corian, …). In DeskProto it is very easy to convert any photo to the relief that is needed for this visual effect. You will see that the results are truly amazing, and that this will allow you to create unique bespoke presents. On the DeskProto website you can view some video tutorials about machining lithophanes.

Creating a Bitmap project

In Lesson 3 you have already seen how you can create a bitmap project. However the image used there was very simple, resulting in a relief with only two Z-levels (black and white). We will now use a digital photo, resulting in a much more detailed relief.

In the file menu select “New Project” >> “New Bitmap Project”. Next press button “Load Bitmap file” (or select that in the File menu) to load the photo that you want to use. We downloaded the image *Meisje_met_de_parel.jpg* (the name of the painting in Dutch) and then reduced the resolution from the original 4095x4794 pixels to 1024x1199 pixels, later-on we will explain why. You can of course use any photo that you like! A rough preview of the photo will be shown on the graphics screen.

By rotating the view you can check that it is a pure 2D picture: one flat plane at constant Z-level. Only the color (gray value) of the pixels in the bitmap varies.

For Bitmap files DeskProto supports BMP, GIF, JPG, PNG and TIF files. Color pictures are automatically converted to gray values (black & white pictures) as for the conversion to Z-levels gray values are needed.

Bitmap Operation parameters

Next you need to set the DeskProto parameters, on two levels: in the Part parameters you define the part, so what you want to machine, and in the Operation parameters you define how you want to machine it. The Operation parameters need to be set first, as for a bitmap these can affect the part size (will be explained below). Don't worry if you get a warning that the part is too large, we will fix that later.

So please now open the Bitmap Operation parameters. The most important parameters are present on the first tab: General. For a lithophane you need a small **Cutter**, or at least one with a small tip, so that the small details in the image can be machined. We used a special cutter for this application: a conic (V-shaped) cutter with a small ball tip. The small tip is needed for the details, the ball shaped tip for a good surface quality, and the conical shape allows to immediately machine at full depth (so without roughing layers). We found a perfect cutter for this application made by Bits & Bits Co. (USA): their model TEB15-020 is a tapered ball end mill with a 0.51 mm (0.02") tip radius.

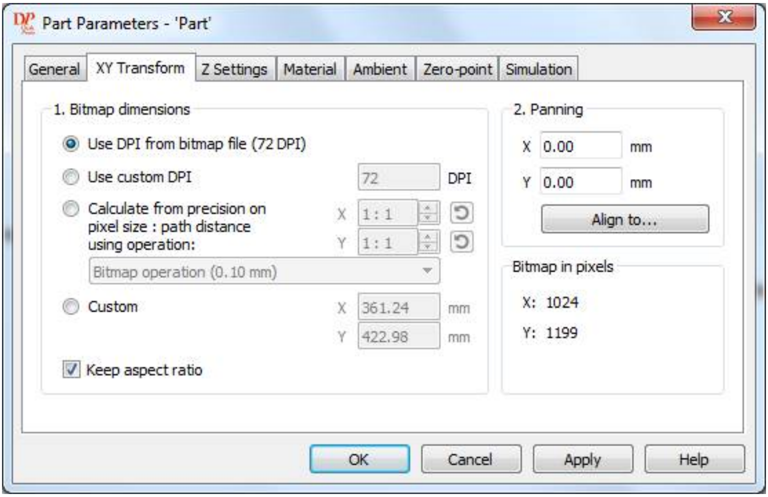
When you cannot find such tapered cutter you can also use a small ballnose, like a 2 mm diameter ballnose cutter. Depending on the cutting length of this cutter it may be needed to apply a roughing operation first.

We selected our tapered ball end mill and the set both the **Distance** between the toolpaths and the **Stepsize** along the toolpath to 0.10 mm (D/33, ca 0.004"). For our machine we left the Feedrate to it's default value, and set the Spindlespeed a bit higher because of the very small tip of the cutter.

Two more further settings in the bitmap operation are needed:
- on tab **Area** we selected "Use bitmap area", as the material block that we want to define (see below) is larger than the bitmap, and only the bitmap area needs to be machined.
- on tab page Movement we set the **Feedrate for high chiploads** on 25%, in order to protect the cutter during the first toolpath (machining at the full width of the cutter).

Part parameters (Bitmap settings)

Next open the Part parameters, and go to tab "XY Transform". The options shown are not the same as in a geometry project: these are the Bitmap settings. As you can see in the image on the next page our bitmap is sized 1024 x 1199 pixels, which at 72 DPI (Dots per Inch) results in 14.22 x 16.66 inch, or 361 x 423 mm). Too large for our small machine, so DeskProto already displayed a Part Size error, and in the tree two error icons are visible.



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

The option “**Calculate from precision**” needs some explanation: it delivers the best results, however it is not easy to understand.
In DeskProto toolpaths are calculated using the Z-grid: a rectangular grid of XY positions, with a Z-value for each position. The Precision (Distance between paths and Stepsize along path, that we have just set to 0.10 mm) sets the size of each grid cell. This Z-grid resembles the bitmap grid of pixels: the bitmap grid also is a rectangular matrix of XY positions, now with a color-value for each position.

The option “Calculate from precision” on scale 1 : 1 makes both grids equal: the bitmap will be scaled to make one pixel exactly match with one cell in the Z-grid. A different ratio can be chosen to align both grids at a different interval. Aligning the grids is important to prevent “**Moire patterns**’: when the grids are not aligned then (for instance) after a series of 10 Z-grid cells with 4 pixels per cell, one cell with 5 pixels will follow, repeating every 10 cells. This will cause visible “ripples” in the resulting relief. Selecting option Calculate from precision (with a proper ratio to achieve the desired size) will prevent this.

When using this option Calculate from precision you need to realize that when you later change the precision, the size of the resulting relief will automatically change as well ! This is why in this lesson the Operation parameters needed to be set before the Part parameters.

For our portrait lithophane we used “Calculate from precision” and kept the ratio to 1:1 (one pixel in one Z-grid cell, so each pixel sized 0.10 mm), which made the Dimensions of the relief 98.52 x 115.36 mm (3.88” x 4.54”).

Attentive readers will notice that this is not what they expected: 1024 pixels at a precision cell size of 0.10 would result in 102.4 mm, not 98.52. This difference is caused by a rounding of the Precision value that is displayed. The actual value for this 1/8” cutter is D / 33 = 3.175 / 33 = 0.09621 mm for one cell in the Z-grid. So also 0.09621 mm per pixel, times 1024 is 98.52 mm.

Panning gives you the option to position the bitmap in 3D space. This will be needed only when you combine a bitmap with a 3D geometry, for now you can just leave it on 0, 0.

On the third Tab page of the Bitmap settings, called “**Z Settings**” you can set the Z-values to be used for the relief. The meaning of the fields Z-value for White and Z-value for Black will be clear after the explanation in the first paragraph of this lesson. Which Z-values need to be used of course depends on the material that you want to use: some experimenting will be needed to find out what material thicknesses will result in ‘black’ and ‘white’ when illuminated from the back.

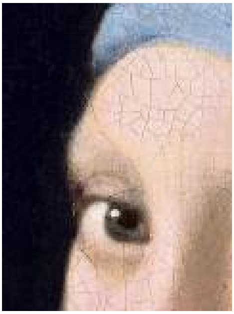
For our lithophane we used a sheet of 3 mm (0.12”) thick ‘Corian’ ® material. We found that the material thickness for the resulting relief needs to be 0.5 mm for white and 2.75 mm for black. So as **Relief depth** we entered -2.5 mm (-0.1”) for white and -0.25 mm (-0,01”) for black.

Finally the size of the **Material block** needs to be set. On tab Material you see that the default option “Use all CAD data” has been selected. Change that to Custom, and enter the size of your material. We used a block of 120 x 140 mm, as said 3 mm thick. This larger size was entered by adding just as much extra on both sides (min and max) for X and for Y: X from -10 to +110, Y from -15 to +125. For Z we simply changed the minimum value from -2.5 to -3.0 mm.

Converting 2D bitmap data to a 3D relief

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

At this point we can explain why we reduced the resolution of the original image for this lithophane project. As said we now used a width of 1024 pixels, and made the pixel size equal to the Precision (size of one Z-grid cell). The first test that we machined was done using the original image (4095 pixels wide), setting the size on “Calculate from precision” on 1:4, which resulted in the same dimensions.



Original image, with cracks in the paint

First lithophane 1:4, thick black lines

Second lithophane 1:1, much better.

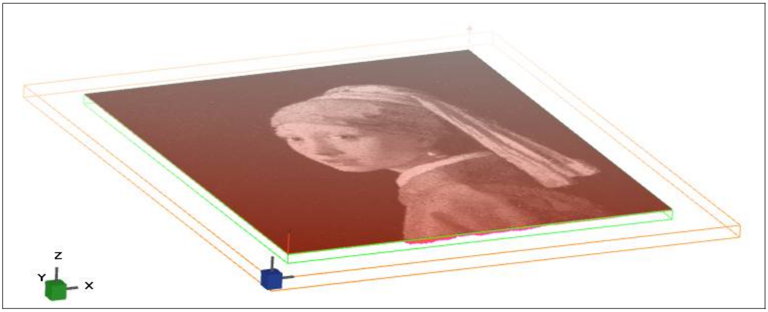
The Vermeer painting is quite old (made in the 17th-century), and so the paint is cracked (“Craquelure”) resulting in a pattern of very thin black lines over the painting. In this first test for each Z-grid cell 16 pixels (4x4) were used. As each Z-grid cell is filled with *the highest Z-value* found within, just one of these 16 pixels being dark (the very thin crack) caused the complete Z-grid cell to have a high Z and thus be dark in the lithophane. See the image above: the crack lines that were very thin in the original image became much thicker in the lithophane, making the result unacceptable. The next version used “Calculate from precision on 1:1” and was much better.

The photo that you use will not be a painting so won’t display Cracquelure, still for any photo a size on scale 1:1 will give the best result in the small details. When you reduce the resolution of an image in a graphics program the new pixel will get some intelligent average of the combined pixels as new color value: better than the highest Z-level that DeskProto applies. Something that we need to work on for a next DeskProto version….

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 that resembles the 3D original. 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.

To the milling machine



You now can calculate the toolpaths, write the NC file and send it to you milling machine in order to mill the lithophane. The WorkPiece zero point is at the left-front corner of the material block, at the top of the block, as is usual in DeskProto.

The image above shows how it looks on the DeskProto screen (for our Vermeer painting). The block of material in orange, with the workpiece zero point in blue. The green lines indicate the are to be machined, which is exactly the bitmap size. Almost all toolpaths are hidden below the bitmap, only on the front (bottom of the portrait) some red toolpath lines are visible.

The resulting lithophane, in 3 mm thick white Corian. Beautiful !

Relief and result

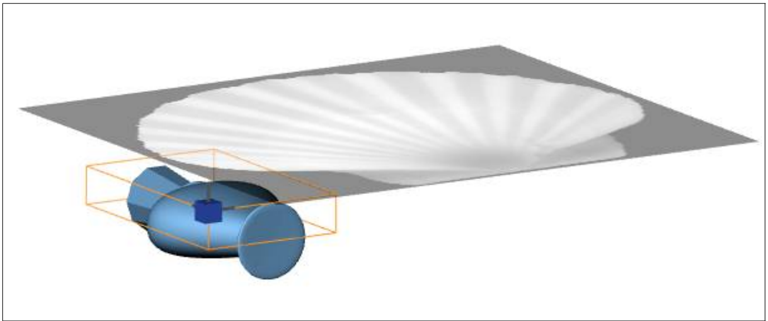
The actual relief that is machined for a lithophane looks very strange. The above lithophane has been machined using exactly the same settings as used in the previous paragraphs. The relief shows deep canyons (grooves) at the left, which become white stripes in the two pullovers, and high mountains in the faces for the dark eyes and dark areas in the mouth. Below the same sheet of plastic is shown, now illuminated from behind. The photo results in a sharper lithophane than the painting in the previous paragraph.

The photo that was used for this lithophane is a stock photo: portrait of a twin brother and sister (source: Nationale Beeldbank / Gertjan Hooijer, 2009).

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 bitmap example project is to create a perfume bottle (also used in lessons 4 and 7) with a shell relief on the side of the bottle. Please do the following:

Start a New Geometry Project
Load the Sample geometry *Bottle.stl* (or *Bottle_inch.stl*)
(the samples can be easily found via the Start Screen)
Load the Sample bitmap file *Shell1.jpg*
Part parameters, geometry settings:
  Rotate the bottle -90 degrees round X
  Set the Material block to Use upper half of geometry.



The image above should be similar to what you now see on your screen. Next step is to correctly scale and position the shell image. That will be easiest when you select a top-view (press the first of the series of cube icons in the toolbar). Now again open the Part parameters, and as Show tab pages for (top of the dialog) select **Bitmap settings**. Open tab page **XY Transform** to scale and position the bitmap.

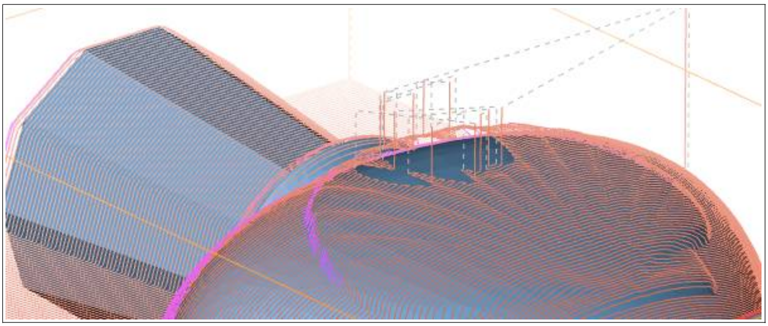
To set the **bitmap dimensions** the option “Calculate from precision” that we used for the lithophane) is not enabled: no bitmap operation is present and thus the precision is not yet available. So you can use either a Custom DPI or a custom size. We applied a custom size of 34 mm for X, resulting in 30.75 mm for Y (in inches 1.3” resp 1.1759”). Press Apply to see the result.

For the bitmap position the **panning** values need to be set. We started with the “Align to...” button: select Center twice, and align with Geometry data. Press OK, and next again press Apply to see the result. For X the position is OK, for Y it is not: the panning value for Y of 31.70 (1.2654”) is too high. Change it as you like, pressing Apply to see the result. We used 22.0 (0.95”) to continue.

Proceed to tab page **Z-settings** to define the relief height. We want to create a positive relief: added on top of the bottle's front surface. So as Z-value for black we set 0.0 (nothing to be added for the pure black background of the image), and as Z-value for white we set 1.5 mm (0.06"). Finally the option "Project bitmap relief on 3D part geometry" on this tab page needs to be checked. When you now close a Part Parameters a message will pop up to warn you that positive Z values are used for the bitmap relief. We will come back on this later.

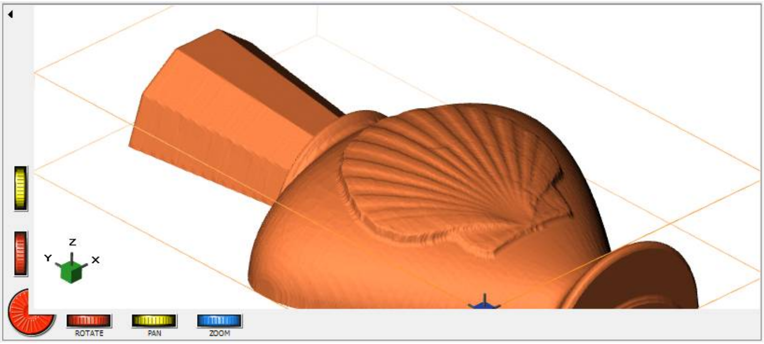
When you now calculate the toolpaths you will see that only the bottle is machined: no shell relief is present. Reason is that in this project only a Geometry Operation is present, where we need a Bitmap Operation. So **add a Bitmap operation** and again calculate toolpaths. Now the shell relief will indeed be visible. So in fact it would have been easier to start this lesson with creating a "New Bitmap project" instead of a new Geometry project. Still for this Tutorial it is important that you learn about the difference. The Geometry operation may not be used as it would remove all material needed for the shell relief (for a negative relief this would not be a problem).

You now see toolpaths for both operations. In most cases the default settings for a Geometry operation and for a Bitmap operation are equal, and then most of the toolpaths will overlap. One difference is the shell relief, and a second difference is hat the bitmap operation will machine a smaller area: the shoulders, top and bottom of the bottle are not machined. Fix this by changing the **Borders** settings in the Bitmap operation parameters: instead of "No extra" (which is OK when machining only the bitmap) you now need to select "Extra for cutter" in order to machine the complete bottle. Now the only difference is the shell relief. You can now delete the Geometry operation (right mouse click on it's line in the tree and select Remove).



One more important problem needs to be fixed: the top of the relief is missing, see the above screenshot. This is what the warning that DeskProto just gave you was about. As Material block we have set the "Upper half of geometry", however on top of the geometry we have added a relief (positive Z-values). As a result the part now is higher than the block of material, and all toolpaths outside the material will be skipped. To fix this problem open the Part parameters, tab Material, and change the selected size from “Use upper half of geometry” to ”Custom”. Now make the value for the maximum Z 1 or 1.5 mm higher (1 mm or 0.04” is sufficient, as the relief is not projected on the highest part of the geometry).

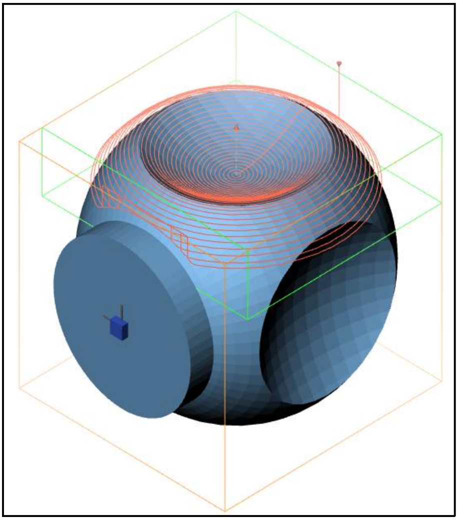
The part to be machined now has been defined, and you can proceed and select the operation parameters that you need: cutter, precision, strategy, etc. You also can decide either to use one operation or to use Roughing and Finishing. Remember that in all cases you need to use Bitmap operations.



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

9: Dice (five-axis machining)

Lesson Nine



The Multi-Axis edition of DeskProto can generate toolpaths for Five-Axis machining. This is quite complicated matter though, and it involves much more work than using the fourth axis (as described in Lesson 5).

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

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

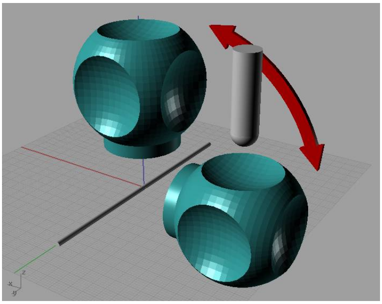
It will be clear that this lesson cannot be applied by users of the Free edition, the Entry edition and the Expert edition.

Five-Axis CNC milling machines

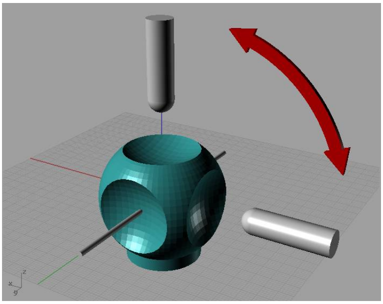
What makes five-axis machining difficult is the fact that so many different configurations of these 5 axes are possible. Machines are available for many of these configurations as well.

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

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



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



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

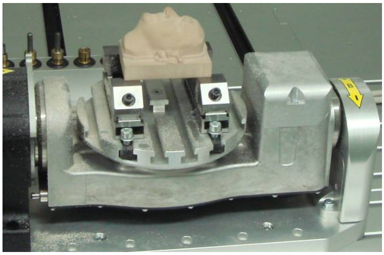
The DeskProto Five-Axis option is meant for **machines where the material block rotates and the cutter orientation remains vertical**. Only rotation axes A (rotation round X) and B (rotation round Y) are supported. Such machines are called **Trunnion-style** 5-axis machines.
It is also needed that the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

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

Distance between rotation axis and geometry

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

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



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

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

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

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

The trick is to set the correct zero point in the STL file.
On the machine shown on the right, the distance between the point of rotation and the clamping plane of the machine vise (in the orientation as shown this is a vertical distance) is 76 mm.
In the file *DentedSphere.stl* the zero point is in the center of the round bottom surface that is clamped on the clamping plane. So an extra translation of 76 mm along Z is needed to get an STL file with the correct zero point for this project.

Easiest is to define this zero point (so to apply this extra translation) in your CAD program and then save a new STL file. If that is not possible then you can do this with DeskProto as well: load the STL file, open the DeskProto Part parameters, tab Zero-point and select None for X, Y and Custom for Z with the required translation. Next save the translated geometry to a new STL file using File > Save Geometry data as, checking the option to include the Translation.

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

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

Configuration of the two rotation axes

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

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

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

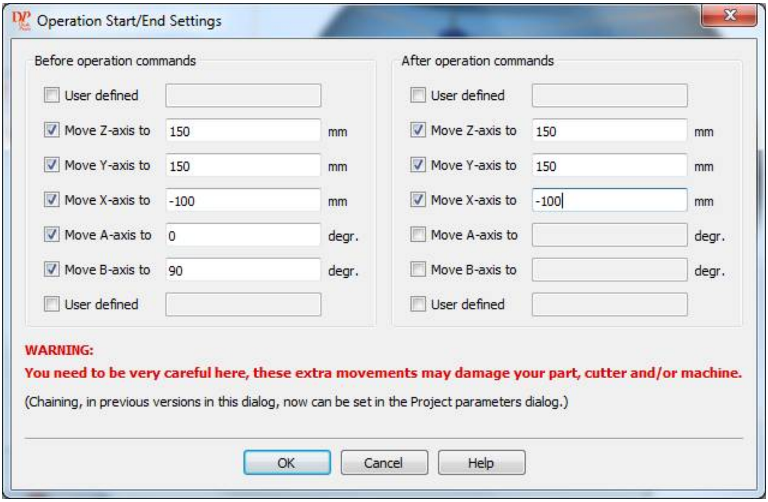
For other machines you may need to experiment to find the correct A and B rotation values to match the rotations round X and Y as set in the Part Parameters. This is for instance true for the Isel machine shown on the previous page, where a rotation of 0.0 degrees for the main rotation axis gives the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

Rotation direction of the two axes

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

Defining the A and B rotation commands

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



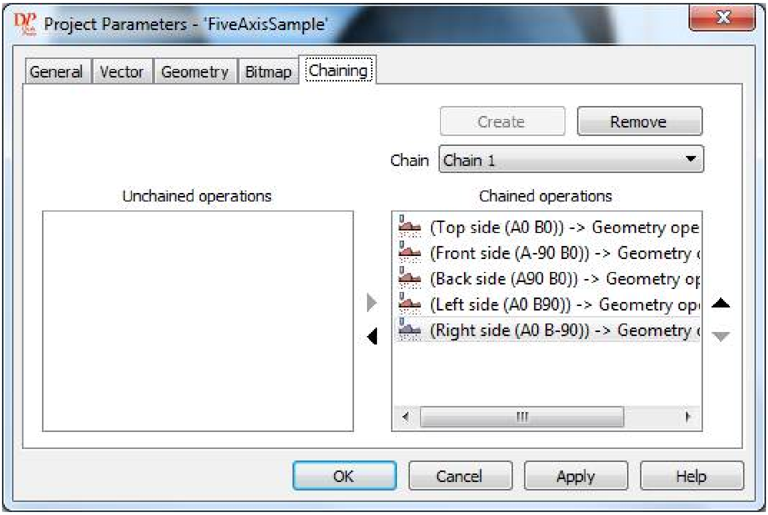
Here a series of commands can be defined to be added to the NC file at the start of this operation (the before commands), and a series to be added at the end (the after commands). In the illustration above, at the start of the operation the A-axis is moved to 0.0 degrees, and the B-axis to 90.00 degrees.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

Very important is that values are entered for X, Y and Z to send the cutter to a **safety retract position** before the rotations start. For many machines this is absolutely needed to prevent a collision between the rotation axis hardware and the cutter. Such collision will damage your part, and perhaps also the cutter and/or the machine. So take care !

The commands in this dialog are executed in the order as shown in the dialog: so first the Z-axis will moves (up), then Y and X will move, and finally the rotations will be done.
Movement commands for A and B can be entered only in case the machine definition used for this part indeed has these two axes defined (tab Advanced settings). If not then these fields will have been grayed out.

Chaining the operations

Finally all operations can be combined in one **Chain**, allowing the export of one combined NC program file. Chaining can be done on the Chaining tab of the Project parameters, see the illustration below.



The illustration shows the situation for sample project *FiveAxisSample.dpj*
You can see that one chain is used, containing all (five) operations. They have been added to the chain using the arrow buttons in the center of the dialog (these buttons are activated after creating a chain). For more information about the use of this dialog please use the Help button.

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

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

Anyway: the result of your own project should be one or more NC files that can be sent to your 5-axis machine. The file(s) will make the machine do the complete job: orient the part, machine one side, rotate to the next orientation, machine, and so on. You can of course also use Roughing and Finishing operations.
When more than one cutter is used, then for a machine with an Automatic Tool Changer (ATC) the result will be one NC file, for other machines several files will be written (a new file after each toolchange).

DeskProto's Five Axis Sample project

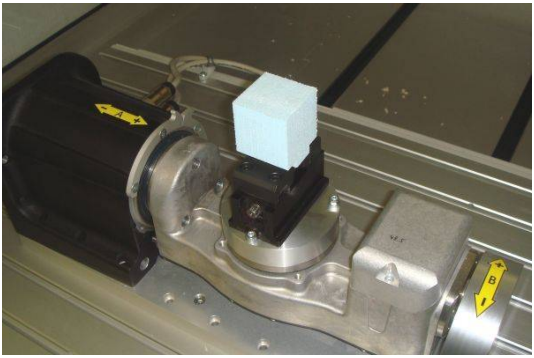
The theory in the preceding paragraphs will now be illustrated by some pictures from the real world. These pictures were made by DeskProto user Robert Zeinecker from Germany, on his Isel Euromod 45 machine with Isel DSH-S rotation axis: thanks !

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

Below we will illustrate the steps in this process.

The first step was to very accurately set the WorkPiece zero point: with the tip of the cutter on the exact spot where the A-axis and the B-axis intersect. This lesson does not show how this is done. You can either measure using a gauge as shown in the photo, or machine a sample cylinder with a roughly set zero point, measure the result and correct the zero point as needed.

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



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

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

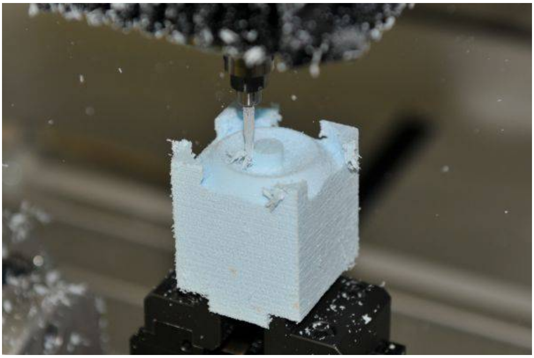
Some experiments were needed to find out which rotations in DeskProto matched which rotations on the machine. For this machine it was needed to reverse the rotation direction for both A and B, as you can see in the table below. As the two axes on the machine have been switched, the X part rotations in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

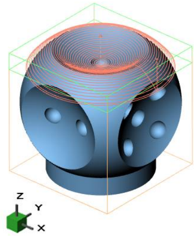
	in DeskProto	on machine:
Part Top	X 0.0, Y 0.0	A 0.0, B 0.0
Part Front	X -90.0, Y 0.0	A 0.0, B 90.0
Part Left	X -90.0, Y 90.0	A -90.0, B 90.0
Part Back	X -90.0, Y 180.0	A -180.0, B 90.0
Part Right	X -90.0, Y 270.0	A -270.0, B 90.0

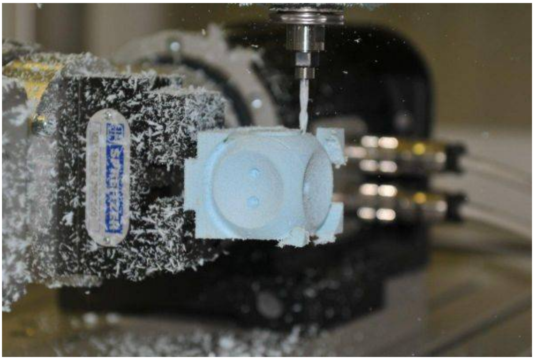
The rotation values for DeskProto have been set as Rotations in the **Part**

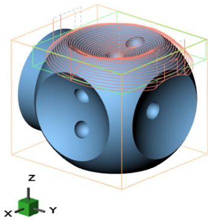
parameters for the five parts, the values for the machine as “Move-to” **Start commands** in the Operation parameters. Here also movement commands for Z and for Y have been used to move the cutter to a **Safety retraction point** before rotating.
As said, the **Translation** has been set to **None**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

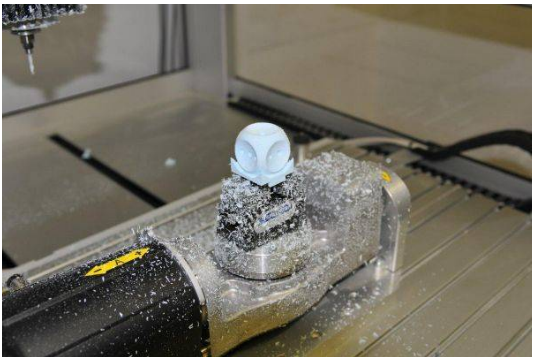
Important to repeat: the rotation values shown are for the machine used in this project: for your machine different values will be needed !

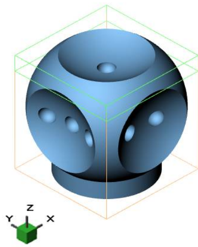












The pictures above show: the milling of sides one (Top), of side three (Left), and the result when all five sides have been completed. Look at the three green “orientator” cubes for the three axis directions in these pictures.

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

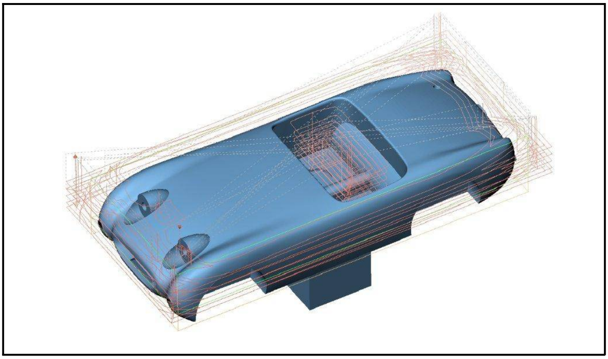
Note 2: Creating a five-axis project is quite complicated and also error-prone (it is easy to forget one of the settings or make a typing error). It may be handy to create a **Template project** (project without geometry) for five-axis machining on your machine. That project should contain all rotation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

A Five Axis Sample car model

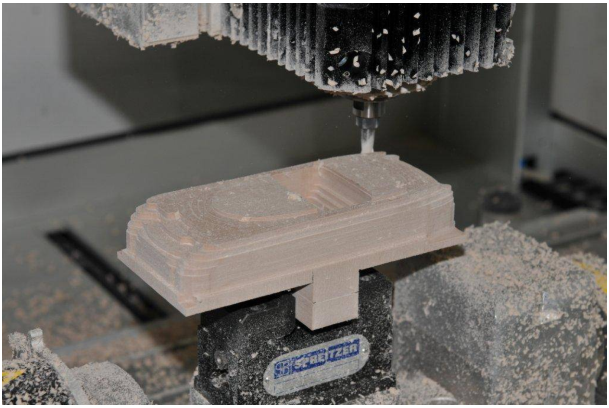
The cube project as described in the previous paragraph was just meant as a teaching help. Once you have mastered this project and have configured DeskProto to work with your five-axis machine, you can of course start with a real project.

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

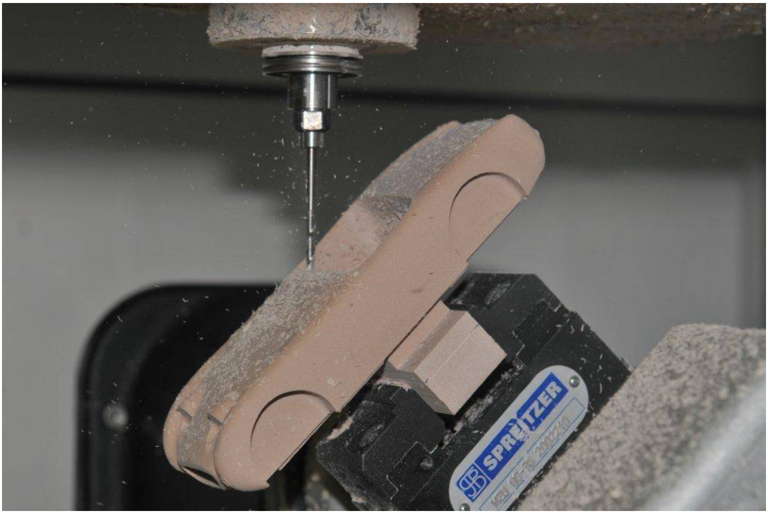
The car body geometry in CAD.



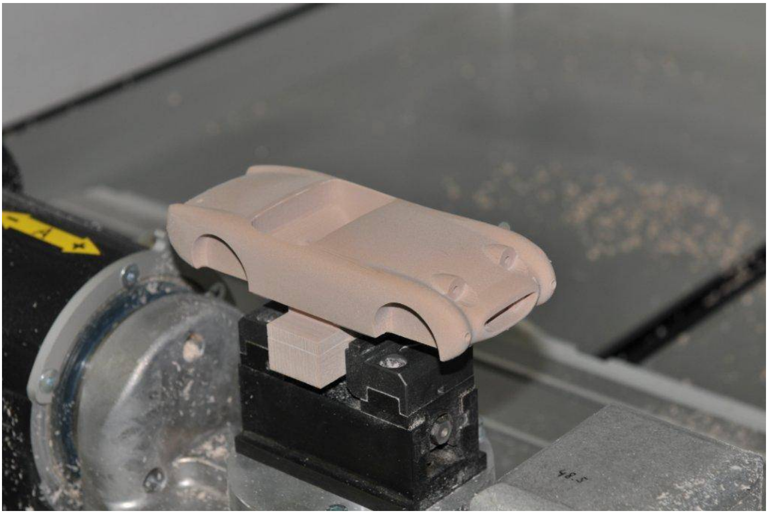
Roughing toolpaths for the top side, in DeskProto.



And the same toolpaths on the machine.



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



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

Index

H

I

L

M

N

O

P

R