Desk*Proto*

Reference Manual

Including Essentials, Screen layout and Command descriptions.

Desk*Proto*

Reference Manual

Including Essentials, Screen layout and Command descriptions.

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

Version 7.0

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

Table of Contents

I Introducing DeskProto

1.1 Disclaimer

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

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

1.2 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.

1.3 Essentials

What does it 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 (DXF files), 3D geometries (STL files), as well as 3D reliefs based on photos (any bitmap files). 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 [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
Starting point for DeskProto is a CAD file (it is not possible to design in DeskProto: CA**M** 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 a MS Windows application, 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 1 GB free disk space: faster/more is better. An OpenGL V3 compatible 3D graphics card is required.

1.4 Specifications

Required operating system:
Windows XP / Vista / Win7 / 8 / 10 or newer (Win 95 / 98 / ME / NT / 2000 are no longer supported). For Win XP Service Pack Three is needed: SP3.

DeskProto supports the 64 bit versions: a special 64 bits build of DeskProto is available to take advantage. Where the standard 32 Windows versions limit the memory use to max 2 GB per application, in 64 bits versions this is almost unlimited (many Terabytes). So for users with large STL files and/or large NC program files this is great news !
The 64 bits build of DeskProto (called x64) will be automatically installed on 64 bits versions of Windows: the Setup will detect that the Windows is 64 bits and install the 64 bits build of DeskProto. Both builds behave identically, you can only find out which version by looking in the About box.

DeskProto is reported to run OK on an Apple Mac under Mac OS X using the Parallels Desktop Windows emulation software.

Required hard disk space:
Minimum 1 GB: about 100 MB for program only, plus at least 900 MB for projects

Required internal memory:
As much as possible, recommended at least 1 GB

Required graphics:
A 3D graphics card (or on-board graphics) that supports OpenGL V3 or newer,

Supported Project files:
DPJ, version 2.0
DPJ, version 3.0 / 3.1
DPJ, version 4.0 / 4.1
DPJ, version 5.0
DPJ, version 6.0 / 6.1
DPJ, version 7.0
Version 1 DPJ files (Windows 3.11) are no longer supported.

Project files exist in two versions: with and without calculated toolpaths.

Supported Vector file types:

DXF AutoCAD Drawing Interchange File
limited to points, lines, polylines, arcs, circles and ellipses (2D)

EPS Encapsulated PostScript and
AI Adobe Illustrator
limited to points, lines, polylines, arcs, circles and ellipses (2D)

Entities in these files that are not supported will be simply ignored by DeskProto.

Supported Geometry file types:

STL STereoLithography
ASCII & binary

DXF AutoCAD Drawing Interchange File
limited to triangles and rectangles (3D)

VRML Virtual Reality Modeling Language
limited to triangles and rectangles

Supported Bitmap file types:

BMP Windows BitMaP
Used by the bitmap operation to load a bitmap

GIF Compuserve Graphic Interchange File
Used by the bitmap operation to load a bitmap

JPG Joint Photographic Experts Group
Used by the bitmap operation to load a bitmap

PNG Portable Network Graphics File
Used by the bitmap operation to load a bitmap

TIFF Tagged Image File Format
Used by the bitmap operation to load a bitmap

Supported NC program files (toolpath data):

NC-program Numeric Controlled
only ASCII, machine-dependent format and file extension

Other supported file-types:

DPW DeskProto Wizard
defines one custom wizard

DPS DeskProto Script
contains a DeskProto script.

DPT DeskProto Toolpath
contains all toolpaths for one DPJ file

Maximum File size

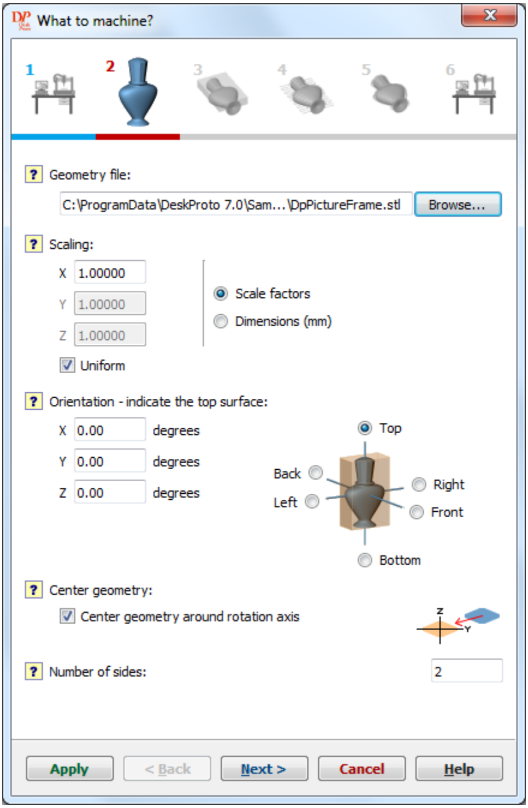
The DeskProto code does not have a maximum for any file. However, because of the memory limitation for **32 bits** Windows versions (see above) the max file size nevertheless is limited under a 32 bits OS. As the memory needs to contain both the geometry and the toolpaths these are related: for a very large STL file (say 1 GB) only simple toolpaths will be possible. Roughly speaking the STL file size maximum for 32 bits DeskProto will be about 800 MB (binary STL).

The **64 bits** version can accept much larger STL files. This mainly depends on how much RAM memory is present: when Windows needs to swap the calculations will become too slow for most users.

1.5 Wizard

For novice users it may be difficult to complete the DeskProto process of generating an NC program file based on your own CAD-data. And for experienced users some complicated models that DeskProto can produce remain difficult to prepare. For these reasons DeskProto features Wizards: a series of dialog screens that guide you step-by-step through the complete process. The DeskProto wizards also tell you where each setting can be found in the 'normal' user interface, to enable you to do it without wizard the next time (if desired). The wizard can be started from the File menu (File >> Start Wizard) and from the Start Screen.

So the DeskProto Wizard is meant to let you execute a task by guiding you through the steps needed. The first wizard screen (see Choosing the type of Wizard) presents a list of the tasks for which a Wizard has been created.



The illustration above shows a typical wizard screen.
On top six icons that indicate the sequence of pages in this wizard. These function as tabs: when colored these icons can be used to navigate to a different page of the wizard by clicking on the icon.
The number of pages of each wizard (and options on each page) depends on which Edition you are running.

Along the bottom the standard Wizard buttons: **Back**, **Next**, **Cancel**, **Help** and (not on the illustration) **Finish**.
The button **Apply** (on the left) will refresh your screen and apply the currently selected settings.
This leftmost button may also show **Calculate**, which will calculate the toolpath with the currently selected settings.

Important are the **yellow question marks** in front of each question: position the cursor over one of these, and DeskProto will show a **Wizard Tooltip** with Help information for this question. Including information where to find this setting in the dialog based user interface.

All functionality offered by the wizards is also available in the normal user interface: the wizards are only meant to make things easier for you, they do not add new options. After finishing any wizard you can fine-tune the settings that the wizard made.
When you open an existing project that was made by the wizard, DeskProto will again use the wizard interface. However after fine-tuning in the dialog based interface that is no longer possible.

In addition a very powerful feature is present called the Custom Wizard: a wizard written in a Script language and later added to DeskProto. This makes it possible for any user and reseller to create a custom wizard for a specific application and/or a specific fixture.

1.6 Support

If you encounter problems while working with DeskProto, please try the following:

1. Search for a solution in the on-line **Help** system. The help is very detailed: every option in every dialog is explained.

2. Look in the **FAQ** (Frequently Asked Questions) on the DeskProto website.

3. Look in the **Forum** on the same website

4. Carefully read the appropriate sections in the printed **Manuals**.

5. In case no solution found: ask the **Dealer** who supplied DeskProto to you.

In case of any other problem: contact Delft Spline Systems.
When you send us a project please do so using the option Create Problem Report: sending the DPJ file is insufficient as that file does not contain the CAD-data nor your cutter/machine/postprocessor definitions.

1.7 Delft Spline Systems

Delft Spline Systems is a Dutch software house, founded in 1984, specialized in the development and the use of CAD/CAM software. The first product of the company, the SIPSURF CAD/CAM package, was released in 1986, ran in MS-Dos, and was specialized in designing freeform surfaces. As for products containing freeform surfaces it is absolutely necessary to create physical models in order to really evaluate the design, a module to easily calculate CNC tool paths has been present from the start.

Since that time Rapid Prototyping has been recognized as vital for product development, has even become a buzz word. Based on a long experience of Rapid Prototyping and NC milling, Delft Spline Systems was able to develop this unique DeskProto package, meant for product designers.

Since that time we have found that DeskProto is very well suited for many other applications as well. Key feature is the ease of use: DeskProto does not aim at CNC specialists (like most other CAM software), but instead at professionals in a different application. Like in jewelry, arts, design, education, prostheses, woodworking, food and sweets, and so on. This is worded in our motto

"CNC machining for non-machinists"

We hope you will enjoy using it, and we expect it will help you in producing high quality designs within a short time.

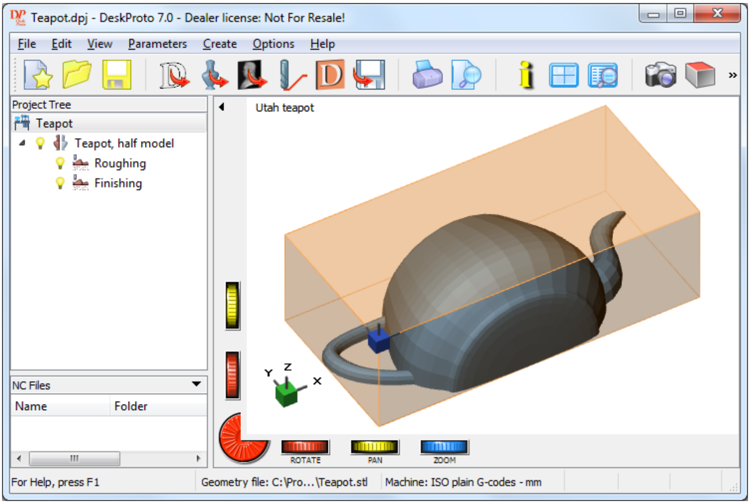
When you want to stay informed about new DeskProto Developments you can subscribe to our **Newsletter emails** by checking that option when downloading a DeskProto trial version from our website.
The news will reach you even more quickly when you follow the DeskProto page on Facebook and/or Google Plus.

Delft Spline Systems
Vogelsanglaan 30
3571 ZM Utrecht
The Netherlands

Tel:	**+31 30 296 5957**
Website:	**www.deskproto.com**

Email **info@deskproto.com**

II Screen layout

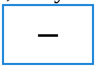
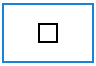
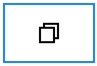


The computer screen presented by DeskProto looks like the screenshot shown above. In this chapter a short explanation will be given of each element on the screen. From top to bottom, the following elements are present:

- The top line is called the **Title bar**. It contains the name of the current project, the word DeskProto, information about your license, and a few Windows icons.
- The **Menu bar** is the next horizontal line. It contains in black characters the names of the available pull-down menus.
- The button bar or **Toolbar** is the horizontal line below, containing a number of buttons for commands that need to be at hand. DeskProto can show these buttons in two sizes (setting in the View menu), the illustration shows the large buttons.
- The **View window**. The large screen-area below the button bar (right part of the screen) is used to display the 3D data in one or more views. The border of this window contains the Thumb-wheels.
- The **Project Tree** window at the left of the view window displays the Project Tree, showing the structure of the current project.
- The **NC Files** window at the bottom left shows the NC Program files that have been saved for this project.

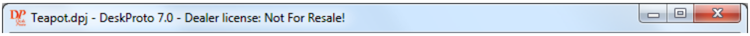
- Finally the bottom line or **Status bar** displays extra information on the DeskProto commands, and some standard Windows messages.

You can **Resize** the DeskProto screen, just as most Windows dialog screens. Two standard sizes and Custom are available:

- Minimized (no window visible, only a button on the Taskbar), to be achieved using minimize button  on the title bar.
- Maximized (full screen window), and in addition, to be achieved using the maximize button  on the title bar.
- Custom size, to be achieved using the restore button  on the title bar. When in custom size mode you can change the size by positioning the cursor on one of the borders or corners of the dialog window (note that the cursor changes to an arrow), pressing the left mouse button and then moving the cursor.

You can also resize two areas of the DeskProto screen: of both the Project Tree window and the NC Files window the size can be changed. You can do so exactly as described above: move your cursor over the border of this screen area, see the cursor change, press the left mouse button and move. This only works on one side of this screen area, called the **splitter**.

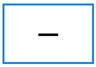
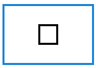
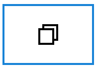
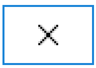
2.1 Title Bar



The Title bar is located along the top of a window.
It's exact appearance depends on your version of Windows and on the Windows Theme that you have selected. In most cases this bar will be semi-transparent, which is a feature of the Windows Aero interface.

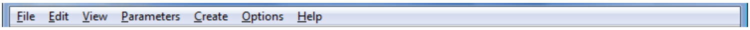
To move the window, drag the title bar. You can also move other dialogs in DeskProto by dragging their title bars.

A title bar may contain the following elements:

- Application Control-menu button (the orange icon "DP")
- Name of the current DeskProto project, or "Untitled" if the project has not yet been saved (here "Teapot.dpj")
- Name of the application (here "DeskProto 7.0")
- License type: commercial, educational, hobby, etc. (here "Dealer license: Not For Resale").
- The Minimize button  to reduce the DeskProto application window to a taskbar button
- The Maximize button  to enlarge the DeskProto application window to fill all available screen space.
- The Restore button  to return the DeskProto application window to its size and position before you chose the Maximize or Minimize command.
- The Close button  to exit DeskProto and close it's window.

The image on the top of this page was made in Windows 7,
the four button images were made in Windows 10.

2.2 Menu Bar

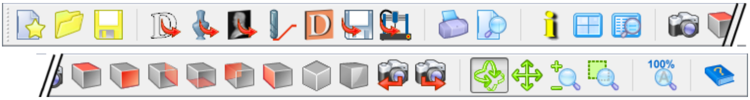


The menu-bar is the main route to all the commands that are available in DeskProto. This is why the command reference in the next chapter of this manual is structured following these pull down menus. Each item in the menu-bar represents a pull down menu, which can be made visible by clicking the left mouse button with the cursor on the item. The following menus are present:

File	File management and Print options.
Edit	Standard Windows options Edit and Properties.
View	Display and View control options.
Parameters	Options to change the parameters for project, part and operations
Create	Toolpath calculation and saving options.
Options	Customization of libraries, defaults and other settings.
Help	Online Help options

Note that in the illustration above one character is underlined for each menu-item. The underlined character can be used to control the program without a mouse. Pressing Alt + F (at the same time) will open the File menu for instance.
In most current versions of Windows the underlining will be invisible: you can press the Alt button to make it visible.

2.3 Toolbar



The toolbar is the series of buttons displayed across the top of the application window, below the menu bar. The toolbar provides quick access to many tools used in DeskProto. All these functions can also be accessed via the menus.

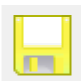
To hide or display the Toolbar, choose Toolbar from the View menu. A check mark appears next to this menu item when the Toolbar is displayed.

The next command in the View menu is "Large Toolbar Buttons", with which you can choose between two sizes of buttons. Again with a check mark if large buttons have been selected (which is the default status). The best choice will depend on the resolution of your screen: on a high-res screen the small buttons will become too small.

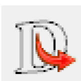
Below a list of all buttons with for each button an explanation of its functions.

Open a New project. Same command as New in the File menu

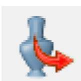
Open an existing project. DeskProto displays the Open dialog, in which you can locate and open the desired DPJ-file.



Save the open project with its current name as DPJ file. If you have not yet named the project, DeskProto displays the Save As dialog.



Load or Add a Vector file into the project.

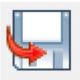


Load or Add a Geometry file into the project.

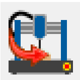
[Load a Bitmap file] into the project (adding not possible: max 1 bitmap file loaded).

Calculate the toolpaths for all visible operations of the current part.

Show a Simulation of the result that you can expect after machining the part.
This is a "toggle button": a button that can be switched on (depressed) and off (normal state), and show's its current state.

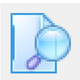


Write the NC program for all visible operations of the current part.



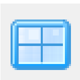
Send toolpaths to machine for all visible operations of the current part.
This button is visible ONLY when this option has been configured (in the Preferences); for some configurations the the button will show a different icon.

Print the image as present in the current view.



Preview how the image would be printed.

Show or hide the Part information dialog.
This is a "toggle button": a button that can be switched on (depressed) and off (normal state), and show's its current state.



Change the Layout out of the views.

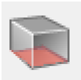
Change which Items should be shown in the active view (the Scene).

Change the Viewpoint (camera position) of the active view.

Set the viewpoint of the active view to Top view (XYZ 0 / 0 / 0).

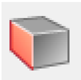
Set the viewpoint of the active view to Front view (-90 / 0 / 0).

Set the viewpoint of the active view to Right side view (-90 / -90 / 0).



Set the viewpoint of the active view to Bottom view (0 / 180 / 0).

Set the viewpoint of the active view to Back view (-90 / 180 / 0).



Set the viewpoint of the active view to Left side view (-90 / 90 / 0).

Set the viewpoint of the active view to Isometric view.

Set the viewpoint of the active view to Default view.

Restore the previous viewpoint settings.

Restore the next viewpoint settings (enabled only after Restore previous view, to undo that restore).

Change mouse-function to rotation.
The four mouse-buttons are "toggle buttons", showing depressed in case selected. De-selecting can be done by pressing one of the other mouse-buttons. Always one of these four buttons is selected.

Change mouse-function to panning.

Change mouse-function to zooming.

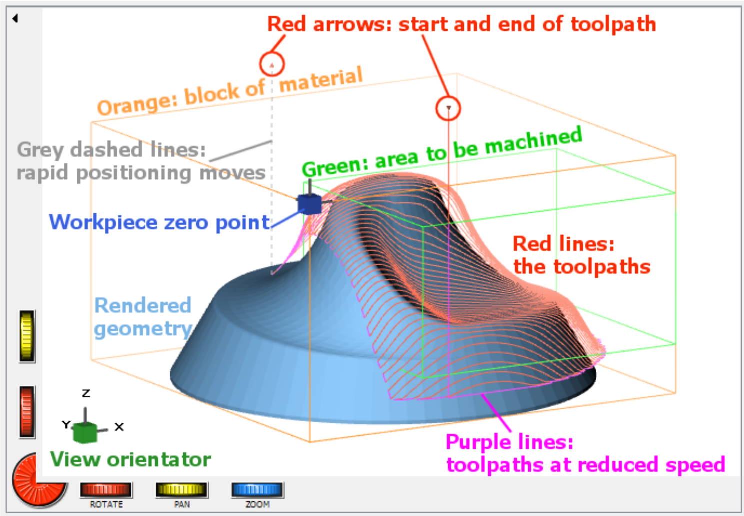
Change mouse-function to zoom window: zooming in by selecting a specific area in the active view.

Note that one of these four mouse-functions is active at any moment, so choosing one means deselecting the previous one. DeskProto also offers other tools: the red thumb-wheels on the screen are for rotations, the yellow for pan and the blue wheel is for zoom. A handy alternative is using the middle mouse button (the mouse-wheel): rotating this wheel zooms, and moving the mouse with the wheel pressed pans.

Set the Zoom percentage to 100% to show the complete geometry.

Display the Help Topics dialog.

2.4 View Window



The view window shows the CAD-data that you are working with, the toolpaths that you create, and much more. In the screenshot above the most important items to be displayed are indicated:

The wireframe block in orange is the **Material block** that you start with for this Part.
The wireframe block in green is the **Area to be machined** for one Operation
Both blocks can be shown shaded by making them Translucent in the **Items Visible** dialog
Drawn in metallic blue is the **Geometry** that you loaded (the STL file). Vector data and bitmaps are displayed in gray.
The lines in red are the **Toolpaths**: the path that the tip of the cutter will follow to create your part.
Start and end of the toolpath are shown as small red arrows.
Some of the toolpaths are in purple: these are done at reduced speed because of a **High chipload**.
And the dashed lines in gray are positioning movements in Rapid.
The green View orientator at the bottom left helps to orient by showing the directions of the axes.
The dark blue orientator shows the position of the WorkPiece zero point.

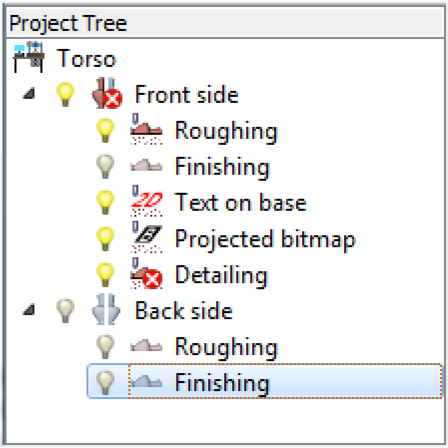
The six **Thumb-Wheels** in the window border can be used to change the camera position.

The list above includes the most used items in the View Window. Many more items can be displayed though: this can be defined using the **Items Visible** dialog. This dialog contains a large number of checkboxes to switch the various items on and off.

Mind the two different orientators: the green Orientator does **not** indicate the WorkPiece zero point: that is shown by the blue Orientator.

Double-clicking in the View window will open the Items visible dialog.
Right-clicking in the View window will make a small menu pop up, called the context menu, offering you a few relevant commands
(right-clicking on a Mac with a one-button mouse can be emulated by pressing the Ctrl (Control) key while clicking the mouse button).

2.5 Project Tree



In DeskProto the **structure** of the Project is shown in the project tree, which is placed on the left of the screen. In the above example the project 'Torso' contains two Parts (icon), one with five Operations and one with two. Three different Operation types can be present: Vector operation (icon), Geometry Operation (icon), and Bitmap operation (icon). When one of these Operations or Parts is in an error status this is shown by the **error icon**.

The **lamp icon** indicates if a line in the Tree is visible (yellow) or not (gray). Only one Part can be current (visible) at a time. Of the Operations in that part none, several or all can be visible. Click on a lamp icon to change the status of that line. The Project line has no lamp icon as that line cannot be turned off.

If you do not see a Project tree window, activate the option Project tree in the View menu by selecting it. The Project tree icon in the menu will become active (drawn in a frame) and the Project Tree will be displayed. De-activating this option (by again selecting it) will make the Tree window disappear. The size of the Window can be changed by dragging it's borders with the mouse.

Shortcuts:

The black arrow button in the border of the View Window opens and closes this Tree Window.

The project tree offers you a number of functions:

Editing parameters
Double-clicking on a line in the tree will open the dialog to edit the parameters of the Project, Part or Operation.

Making a part current
To see a particular part (in case you have defined more than one part) you should make it current. To make a part current just click with the left mouse-button on the gray lamp icon of that part. You can also use the Context menu (see below). When a part is not current, its icon is grayed.
There is always exactly one part that's current. No more, no less.
You cannot make a Part "un-current" by clicking on a yellow lamp icon: you can only make a different part current.

Making an operation (in)visible
To be able to see how the toolpaths look for a particular operation, this operation should be visible. To make an operation visible just click with the left mouse-button in the tree on the gray lamp icon of that operation to make it yellow (turn the light on). Clicking on the yellow lamp icon of a visible project will make it invisible. When an operation is not visible, its icon is grayed.
Of the operations in a Part none, one, several or all can be visible.

Displaying parameters in the status bar
When you single-click on one of the items in the project-tree, that particular item is highlighted (shown with a blue background), meaning that it is **selected**. At that moment some of the parameters of that item will be shown in the Status bar.

Context menus
When you **right-click** on an item in the tree a small menu pops up called the context menu, offering you a number of functions (right-clicking on a Mac with a one-button mouse can be emulated by pressing the Ctrl (Control) key while clicking the mouse button).
The available functions will be different for each line of the tree, and will include the following options:

Edit the parameters of that tree-item.

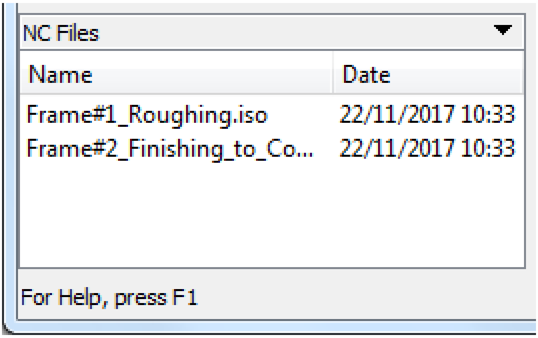
Add a Part to the project. The settings of the default part are used.
Add an Operation to that part. The settings of the first default operation are used. There are Add options for a Vector Operation, a Geometry Operation and a Bitmap Operation.
Copy an operation or a part will add an item that is identical to the current one.

Remove a Part from the project. This is only possible when there will be at least one part left after it has been removed.
Remove an Operation from that part. This is only possible when there will be at least one operation left in the part where it belongs to after it has been removed.

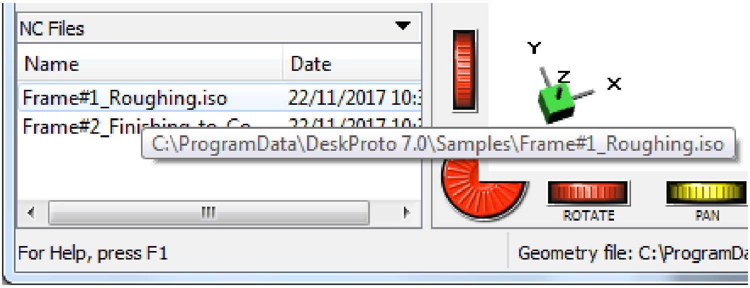
Move Parts and Operations in the Tree can be used when the sequence is important (for instance first roughing, then finishing).

The other options in the context menu will be self explaining.

2.6 NC Files



The NC files window shows a list of NC program files that have been saved for this project. For each file its Name and Date are listed. This may be easy for you to manage NC files for this project.



Hoovering over the filename (positioning the cursor there without clicking) will make DeskProto show the complete file specification of the NC file in a tooltip: see the screenshot above.

Double-clicking a file will open the file using the default program that has been set for that file-type in File Explorer. For instance double-clicking a .TXT file will open it in Notepad. You can for instance configure your machine's control software to open your NC files, or a text editor in case you want to check the file's contents.

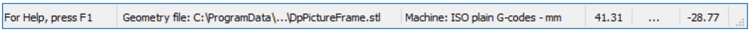
By right-clicking on any line in this window you can open a Context menu, offering the following options:

- Remove the file-name from the list

- Delete the file
- Open the file - same effect as the double-click just described
- Open the file location - which will show the NC program in File Explorer
- Send the file to your machine (only in case that option has been configured)
- Add a file - for instance to add a TXT file with your own project documentation.

The NC Files window can made visible or invisible by checking or unchecking the option NC Files list in the View menu. This same effect can be achieved by pressing the black arrow button in the title bar of this window. Note that the NC Files window can only be visible when the Project Tree Window is visible too.

2.7 Status Bar



The status bar is displayed at the bottom of the DeskProto window, and gives status information about various relevant items. To display or hide the status bar, use the Status Bar command in the View menu. A check mark appears next to the menu item when the Status Bar is displayed.

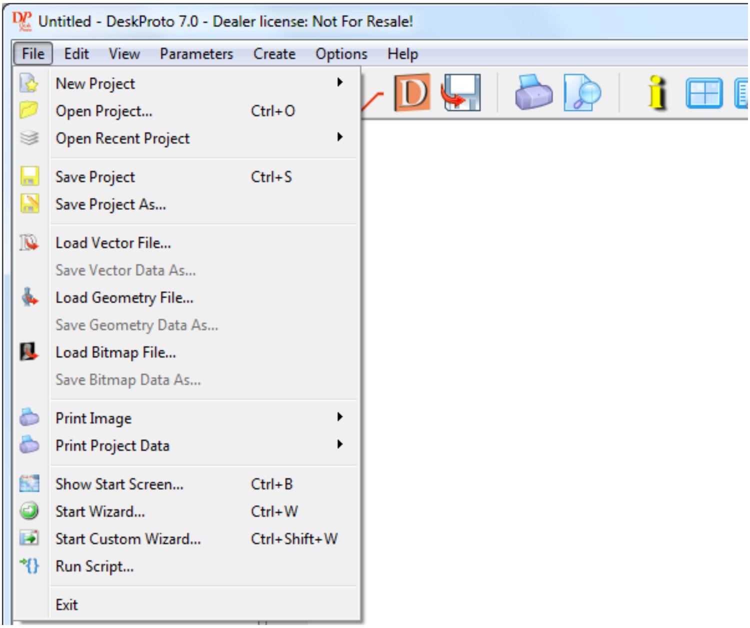
The left side of the status bar describes the actions of toolbar buttons as you point to them (a sort of extra Help information). Menu item actions are also described when you use the arrow keys to navigate through the menus.

In the middle area the most important parameters of the selected tree-item are displayed: CAD filename and machine for the Project, dimensions for a Part, cutter and precision for an Operation.

At the right side of the status bar the **coordinates of the current mouse position** are displayed, however only when the geometry is displayed in one of the main views. These coordinate values shown are in “Translated” coordinates: the coordinates as used in the NC file. This is a very handy option that enables you to quickly check dimensions and positions on the screen.

III Menu commands

3.1 File Menu



The File menu contains all options for File management and for Printing, conform Windows conventions.
On the right side of the menu you can see "shortcuts" for a number of commands: for instance "Ctrl+N" means to keep the Ctrl key pressed and then press the O key: DeskProto will perform the Open Project command. No need to use your mouse !

Note that not all options are present in the DeskProto Free edition and Entry edition.

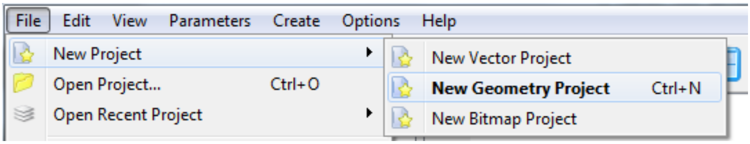
3.1.1 New Project

This File menu command creates a new project in DeskProto.
This project will be called "Untitled" until it has been saved.
You can select one of three project types: Vector project, Geometry project, Bitmap project,
by selecting one in a sub-menu.

The difference is the type of operation that will be present:

- a **Vector project** features a Vector operation, and is meant to process Vector Data (like a 2D DXF file)
- a **Geometry project** features a Geometry operation, and is meant to process Geometry Data (like a 3D STL file)
- a **Bitmap project** features a Bitmap operation, and is meant to process Bitmap Data (like a JPG file)

Combined projects are possible by adding operations once the project has been opened.



One of these three project types is the **default, shown in bold characters**.
You have selected this default when you first started DeskProto, in the Initial Settings dialog.
It can be changed later by editing the default Part (Options menu) and changing the type of operation that is present.

When starting DeskProto automatically opens a new default project, so no need to again use "New Project" at program start.
As a next step you can then load the CAD file(s) you want to use: Vector data, Geometry data and/or Bitmap data.

You can open an existing project with the Open Project command.

Shortcuts:

Toolbar: 
Keys: CTRL+N

3.1.2 Open Project

This File menu command opens an existing project. The currently open project will be closed, and the standard Windows Open File dialog will be displayed, showing all DeskProto project files (.DPJ).

You can't open more than one project at a time.
You can create a new project with the New command.
A 'Close' option is not present: you can close your current project using either 'New', 'Open' or 'Exit'.

Important to know is that you can not load a CAD Data file using the Open command: it can only open DeskProto projects. In order to load a vector file, geometry or bitmap file you need to use the command Load Vector File, Load Geometry File or Load Bitmap File. Even easier is to use one of the Wizards.

Shortcuts:

Toolbar: 
Keys: Ctrl+O

When opening a project file, the CAD file(s) that were used will be read again: the DPJ files does not store the CAD-data but only links to the external CAD files. The CAD file must thus be found at the same place as where it was when the project file was saved. So when sending out a project file make sure to also send all CAD files.

If the CAD file cannot be found (for instance when the DPJ file was copied from a different computer system) then DeskProto will check if a CAD file with the correct name can be found in the current directory (same as the DPJ file) or in the default Data directory. If yes, DeskProto will ask you if it can use that file instead, see the File Not Found dialog.

3.1.3 Open Recent Project

This File menu command will show a list containing the latest projects that you have opened (maximum 20).
Click on any project in the list to again open it.

This list is called the **Recent File List**, and also MRU which stands for Most Recently Used.
It is a convenient standard option in current software.

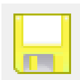
3.1.4 Save Project

This File menu command saves the open project to its current name and directory, in a DeskProto project file (.DPJ). When you save a project for the first time, DeskProto will display the Save As dialog so you can name your project.

If you want to change the name and directory of the open project before you save it, choose the Save As command.
Also if you want to save the DPJ file with toolpaths you need to use that command.

Note that the project file does not contain the CAD-data, only a link to the CAD-file. For saving the geometry after it has been changed use the Save Geometry As command.

Shortcuts:

Toolbar: 
Keys: Ctrl+S

3.1.5 Save Project as

This File menu command saves and names (or renames) the open project, using the Save As dialog.

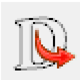
In this dialog you can:
- Name your project.
- Select a new location for the project file.
- Select the type of DPJ file to be written: standard or with Toolpaths.

To save a project with its existing name and directory, use the Save command.

3.1.6 Load/Add Vector File

The File menu command **Load Vector File** displays the standard Windows Open File dialog in which you can select the vector file you want to load. It will be showing all Vector file types that DeskProto can read: DXF, AI and EPS.
After browsing the file it's contents will be loaded, at project level (so accessible for all parts and for all operations).

Shortcuts:

Toolbar: 

When you already have loaded vector data, this command will change to **Add Vector File** and the new vector curves will be Added. If you want to replace your current Vector file with a new, then go to the Parameters menu, choose the option Project Parameters and go to tab Vector.

Only when Adding a second vector file an extra dialog will pop up: the Vector Data Transformation dialog. This dialog enables you to position the new vector curves relative to the current collection.

3.1.7 Save Vector Data as

This File menu command displays the standard Windows Save File dialog in which you can define the vector file you want to write.
This command makes it possible to:
1- save vector data with one or more Transformations applied
2- merge several vector files to one combined new file.
3- NOT YET IMPLEMENTED - save vector data in a different format (so use DeskProto as a converter for vector files).

Of course this is only possible in case you have previously loaded Vector data.

About 1:

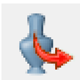
Before the standard Save dialog an extra dialog will pop up: the Save Vector Data Options dialog, which enables you to apply Transformations that you have set in the Part Parameters.

About 3:
For importing Vector files DeskProto supports three file formats: DXF, EPS and AI.
However, for exporting Vector files any DXF is supported.

3.1.8 Load/Add Geometry File

The File menu command **Load Geometry File** displays the standard Windows Open File dialog in which you can select the geometry file you want to load. It will be showing all Geometry file types that DeskProto can read: STL, DXF and WRL.
After browsing the file it's contents will be loaded, at project level (so accessible for all parts and for all operations).

Shortcuts:

Toolbar: 

When you already have loaded a geometry, this command will change to **Add Geometry** and the new geometry will be Added, to form one combined new geometry. If you want to replace your current geometry with a new, then go to the Parameters menu choose the option Project Parameters and go to tab Geometry.

Only when Adding a second geometry file an extra dialog will pop up: the Geometry Transformation dialog. This dialog enables you to position the new geometry relative to the current one.

3.1.9 Save Geometry Data as

This File menu command displays the standard Windows Save File dialog in which you can define the geometry file you want to write.

This command makes it possible to:
1- save geometry data with one or more Transformations applied
2- merge several geometry files to one combined new file.
3- save geometry data in a different format (so use DeskProto as a converter for geometry files).

Of course this is only possible in case you have previously loaded Geometry data.

About 1:
Before the standard Save dialog an extra dialog will pop up: the Save Geometry Data Options dialog, which enables you to apply Transformations that you have set in the Part Parameters.

About 3:
DeskProto supports six file formats for geometry files, you can select one in the **Save as Type** box.

STL	STereoLithography File	ASCII
STL		Binary
DXF	AutoCAD Drawing eXchange File	Polyface Meshes
DXF		3D Faces
VRML	Virtual Reality Modeling Language	Version 1.0
VRML		Version 2.0

STL is preferred as being most standard, and then binary results in a much smaller file size than ASCII. So Binary STL is the default file type here

3.1.10 Load Bitmap File

The File menu command **Load Bitmap File** displays the standard Windows Open File dialog in which you can select the bitmap file you want to load. It will be showing all Bitmap file types that DeskProto can read: BMP, JPG, GIF, PNG and TIF.
After browsing the file it's contents will be loaded, at project level (so accessible for all parts and for all operations).

Shortcuts:

Toolbar: 

DeskProto supports only one open bitmap file per project. So adding a second bitmap file (like for geometry files and vector files) is not possible.
Reason is that geometry data and vector data comes with a position in 3D space, so combining several files can make sense.
Bitmap files do not have such position in space.

If needed you can combine several bitmap files to one large image (using any graphics program) and import that new image file in DeskProto.

3.1.11 Save Bitmap Data as

This File menu command displays the standard Windows Save File dialog in which you can define the bitmap file you want to write.
This command makes it possible to save bitmap data in a different format (so use DeskProto as a converter for bitmap files).

Of course this is only possible in case you have previously loaded Bitmap data.

DeskProto supports four file formats for exporting bitmap files: BMP, JPG, PNG and TIFF. You can select one in the **Save as Type** box.

3.1.12 Print Image

This File menu command prints an image of the DeskProto View Window.

This command presents a Print dialog, where you may specify the range of pages to be printed, the number of copies, the destination printer, and other printer setup options.

Shortcuts:

Toolbar: 
Keys: CTRL+P

3.1.12.1 Print Preview

This File menu command displays the image of the page to be printed.
Two versions of this command are present in DeskProto:

- Print Image Preview, which shows the View Window as it would appear when printed.
- Print Project Data Preview, which shows the Project Data as it would appear when printed.

When you choose this command, a print preview window will pop up in which the page will be displayed in their printed format. The print preview toolbar offers you options to view either one or two pages at a time; move back and forth through the document; zoom in and out of pages; and initiate a print job.

Shortcuts:

Toolbar:  (for Print Image Preview)

3.1.12.2 Print Page Setup

This File menu command selects a printer and a printer connection.
Two versions of this command are present in DeskProto:

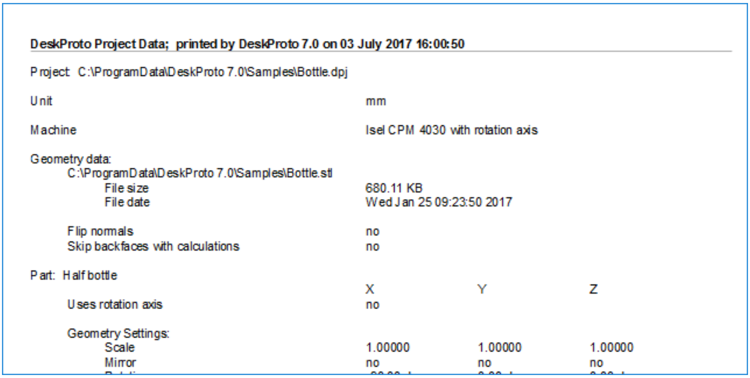
- Print Image Page Setup, for printing the View Window.
- Print Project Data Page Setup, for printing the Project Data.

This command presents the standard Windows Print Setup dialog, where you specify the printer and its connection. The available options will depend on the Printer driver that you have selected.

Note that some CNC milling machines also use a Windows Printer Driver and thus a Print Setup Dialog. Do NOT select such milling machine here, instead use the DeskProto Preferences. This command is only for printing on paper.

3.1.13 Print Project Data

This File menu command prints all the data in the project, that means all the project parameters, the parameters of all of its parts, and the parameters of all of their operations. See the example image below.



DeskProto Project Data; printed by DeskProto 7.0 on 03 July 2017 16:00:50

Project C:\ProgramData\DeskProto 7.0\Samples\Bottle.dpj

Unit	mm		
Machine	Isel CPM 4030 with rotation axis		
Geometry data:			
C:\ProgramData\DeskProto 7.0\Samples\Bottle.stl			
File size	680.11 KB		
File date	Wed Jan 25 09:23:50 2017		
Flip normals	no		
Skip backfaces with calculations	no		
Part: Half bottle			
	X	Y	Z
Uses rotation axis	no		
Geometry Settings:			
Scale	1.00000	1.00000	1.00000
Mirror	no	no	no

The printed pages can be used for backup and documentation purposes.

This command presents a Print dialog, where you may specify the range of pages to be printed, the number of copies, the destination printer, and other printer setup options.

3.1.13.1 Print Preview

This File menu command displays the image of the page to be printed.
Two versions of this command are present in DeskProto:

- Print Image Preview, which shows the View Window as it would appear when printed.
- Print Project Data Preview, which shows the Project Data as it would appear when printed.

When you choose this command, a print preview window will pop up in which the page will be displayed in their printed format. The print preview toolbar offers you options to view either one or two pages at a time; move back and forth through the document; zoom in and out of pages; and initiate a print job.

Shortcuts:

Toolbar:  (for Print Image Preview)

3.1.13.2 Print Page Setup

This File menu command selects a printer and a printer connection.
Two versions of this command are present in DeskProto:

- Print Image Page Setup, for printing the View Window.
- Print Project Data Page Setup, for printing the Project Data.

This command presents the standard Windows Print Setup dialog, where you specify the printer and its connection. The available options will depend on the Printer driver that you have selected.

Note that some CNC milling machines also use a Windows Printer Driver and thus a Print Setup Dialog. Do NOT select such milling machine here, instead use the DeskProto Preferences. This command is only for printing on paper.

3.1.14 Show Start Screen

The DeskProto Start Screen shown above is a help to quickly start the task that you need.
You can open the Start Screen via the File menu (File >> Show Start Screen).
A keyboard shortcut is available as well: Ctrl + B (B for "Begin", as the S already is used).

It will first show the Edition that your are running,
next your Name and Location (as owner of the license),
and the type of License, with the restrictions that apply (of any).

Three groups of tasks are presented:
- Open recent project
- Start new project
- Other tasks

Each line in these three lists is a link that will directly start that task.

Recent projects will be available only when you have used DeskProto V7 before. In this dialog only the latest 5 projects will be shown. Command Open will allow you to open any project file by browsing one.

Starting a New project can be done either using the Wizard interface or the Dialog based interface.
Default choice for this dialog is to start a New project using the Wizard: that is what will happen when you simply press the Enter key (indicated by the icon at this line). This is the most convenient option for novice users.
The difference between Vector/Geometry/Bitmap projects is explained on page New project.

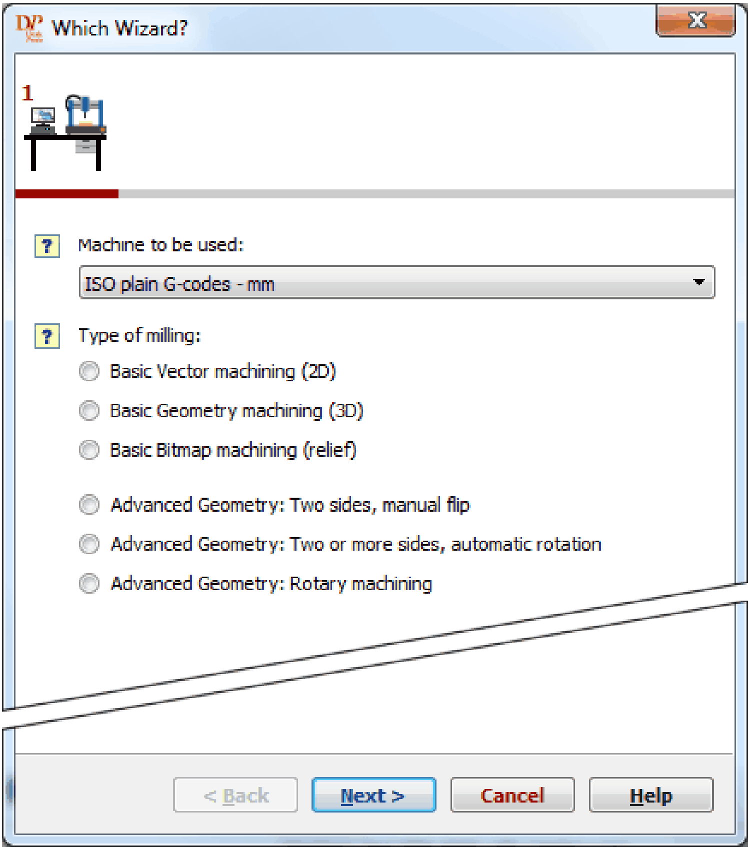
The DeskProto Setup has installed a number of **Sample projects** and **Sample geometries** on your PC. Including some great geometries like the DeskProto picture frame: see the lessons in the DeskProto Tutorial book.

Conform Microsoft's specifications the **Samples** have been installed in the \ProgramData\ folder, which may not be easy to find as it is a hidden folder and its location is different per Windows version. Sorry about that, unfortunately for standard users other locations are not permitted by Microsoft.
Using the checkbox **Use samples folder** you will always be able to find the sample geometries.

In the **Other tasks** section, **Open sample project** makes it easy to find the sample projects.
The other three 'other tasks' require an Internet connection, as they will start your browser and open a web page.

The checkbox **Show this start screen on startup** makes this screen automatically appear at each start of DeskProto. After having de-selected this option, you can still access the Start Screen using the command **Show Start screen** in the File menu.

3.1.15 Start Wizard



The Wizard-based user interface is an important feature of DeskProto. It makes it possible for users without CAM know-how to easily create the toolpaths that they need for their projects. Each wizard is a series of dialog screens (forms) that need to be filled in: the wizard in a way *'takes your hand'* and guides you through the process of creating toolpaths.

When you create a project using a wizard and you save the project without making any changes after finishing the wizard, on re-opening DeskProto will show the project using the same wizard. So it is possible to use DeskProto seeing only the wizard interface. This is new in DeskProto V7, and will be a great help for inexperienced users.

Navigating the Wizard is done using the Wizard buttons, on the bottom of the screen:

Back and 

Next let you navigate the wizard pages,

Cancel and 

Help are standard buttons. Every wizard page has it's own Help information page.

Finish will replace Next on the final page of the Wizard,

Apply will refresh your screen and apply the currently selected settings,

Calculate will calculate the toolpath with the currently selected settings.

Navigation icons: The DeskProto wizards start with the page shown above, called "Which Wizard?". On the top of the page you can see one icon, after selecting one of the wizards a series of icons will be shown: one for each wizard page. At this point these extra icons are all gray, when progressing they will be colored: then they can be used as tab-pages to navigate (click on a colored icon to jump to that page of the wizard).

Help information: Important are the 

yellow question marks in front of each question. When you position the cursor over such mark a Tooltip will pop up, giving extra information about that question. It will also tell you where to find that setting in the Dialog based interface.

The first question, "*Machine to be used*", in most cases needs not be changed as your default machine will already have been selected here.

The second question, "*Type of milling*", presents the six wizards that you can choose from: you need to select one before continuing.
Not all wizards will be available for all users (some may have been grayed out): some wizards are not available in all DeskProto **Editions**, and some wizards are only available in case you selected a machine with a rotation axis.

Also: in the Free edition and the Entry edition the available wizards do not include all described options.

Basic Vector Machining
This wizard creates 2D toolpaths for just one Vector file (DXF, EPS, AI). It is meant for novice DeskProto users, and explains this procedure step-by-step. You can choose to use either Profiling or Pocketing toolpaths.

Basic Geometry Machining
This wizard creates 3D toolpaths for just one Geometry file (STL, DXF). It is meant for novice DeskProto users, and explains step-by-step the procedure to create an NC toolpath file (NC program) based on your geometry. The model will be machined from one side, using three operations: Roughing, Finishing and a smoothing contour.

Basic Bitmap Machining
This wizard creates 3D toolpaths for a relief based on a Bitmap file (BMP, GIF, JPG, PNG, TIFF file). It is meant for novice DeskProto users, and explains this procedure step-by-step.

Advanced Geometry (the three advanced wizards are all for geometry machining):

Two sides, manual flip, also called **Two-sided Wizard** (not available in the Free Edition and the Entry edition)
This advanced wizard is a unique feature of DeskProto, and makes it very easy for you to create a compete 3D part by machining it from two sides, on any three axis milling machine. DeskProto assists you by taking care of the repositioning needed to machine the second half: no need to change the workpiece zero point (starting position of the cutter).

Two or more sides, automatic rotation, also called **N-Sided Wizard** (available only in the Multi-Axis edition)
N-sided milling is meant for machines with a rotation axis (A-axis), and allows **indexed machining**: this wizard generates toolpaths to machine the part from several (N) sides, with a rotation in-between. The number of sides can be freely chosen: for two sides the result is the same as for the previous wizard, though now with automatic rotation.

Rotary Machining (available only in the Multi-Axis edition)
If your machine is equipped with a rotation-axis (A-axis) you can use this wizard to create toolpaths for a model that is machined from all sides:

rotation axis machining. In contrast to the previous wizard, now the material rotates during machining.
A rotation axis is an extra piece of equipment that lets your model rotate during machining (just like the meat rotating on a spit above a barbecue).

Note that all functionality offered by the wizards is also available in the dialog-based user interface: the wizards are only meant to make things easier for you, they do not add new options. After finishing any wizard you can still use the dialogs to fine-tune the settings that the wizard made.

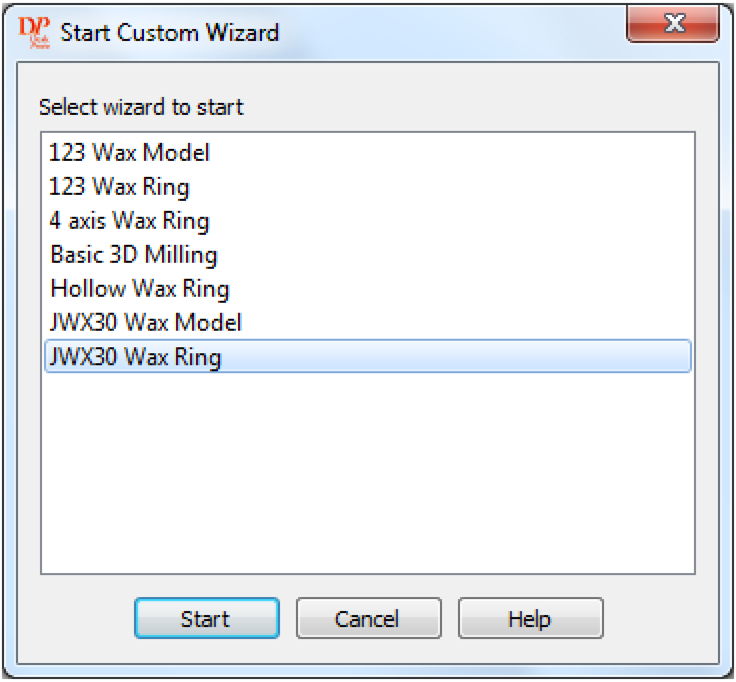
You can find the Wizard in the File menu (File >> Start wizard) or in the Start Screen.
A keyboard shortcut is available as well: Ctrl + W.

3.1.16 Start Custom Wizard

The Custom Wizard (also called Script wizard) is a wizard written in a Script language and is stored as a separate file (so not in the file DeskProto.exe).
This is a very powerful option, as it makes it possible for any user and reseller to create a custom wizard for a specific application and/or a specific fixture.

Using a Custom Wizard.

Script Wizards can be accessed via the File menu.
A keyboard shortcut is available as well: Ctrl + Shift + W.



After selecting command Start Custom Wizard the dialog shown above will pop up, allowing you to select which Custom wizard to use. This list is filled when DeskProto starts, with all valid Wizards found in the subdirectory \Wizards\ of the DeskProto directory, for instance in C:\Program Files\DeskProto 7.0\Wizards\
As most of the wizards shown in the illustration above require a rotation axis, the list of wizards will be much shorter for users of a Free edition, Entry edition and/or an Expert edition of DeskProto.

A Custom wizard can also be started by:
- calling it's DPW-file as a command line parameter.
- double-clicking it's DPW-file in Windows explorer.

When you are a user of a DeskProto Custom Wizard and need any Help information, please refer to the documentation that came with that Wizard.

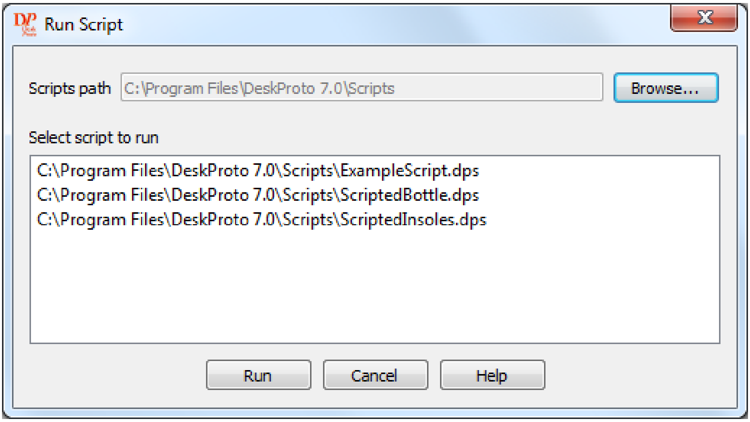
More information about Custom wizards can be found on the Custom Wizard page.

3.1.17 Run Script

Normally when you use DeskProto you will load a CAD-file, set a number of parameters, calculate toolpaths and save an NC program file. So a number of actions that are manually done, in a specific sequence. In some applications the actions to be done and the parameter settings to be used are always the same, which makes it possible to automate this task. Such automation can be achieved by Scripting.

A Script is a list of actions to be done, saved in a file. These actions of course need to be described in a language that the computer will understand: in a DeskProto script you need to use the **JavaScript** scripting languages. For more information also see the Scripts page.

You can start a script via the "Run Script..." command the File menu.
A keyboard shortcut is available as well: Ctrl + R.



This command will open the Run Script dialog as shown above. Here you can browse the Script file (files are seen with extension **.dps** for **D**esk**P**roto **S**cript file) and then press the Run button to start it.

Complete automation can be achieved by starting DeskProto with the name of the Script file as command line parameter. Then this complete dialog will be skipped: the script will be started and executed automatically. More information about this in Command line parameters.

Note that also a special type of Script does exist, called the Script Wizard, which does exactly as the name suggests. These script wizards can not be started using this Run Script command.

```
// test functions
if (loadGeometry())
        addPart();

// returns true if geometry was successfully loaded
function loadGeometry()
{
        // empty string as path will show an open file dialog
        result = DeskProto.project.loadGeometry(true, "");
        return result;
}

// creates a new part, changes some parameters and calculates it
function addPart()
{
        // copy current active part, the copied part will be the new ac
        DeskProto.project.copyPart();
        // change name
        DeskProto.project.activePart.name = "This Part has just been ad
        // rotate -90 degrees around x axis
        DeskProto.project.activePart.setRotation( -90.0, 0.0, 0.0 );
        // calculate toolpaths
        DeskProto.project.calculateToolpaths();
}
```

Above you see the very simple script file ***ExampleScript.dps*** that was present in the dialog just shown. It contains (and calls) two functions, one to load a 3D geometry file, and one to add and rotate a part. Line starting with "//" in JavaScript are comment lines. This is just a very simple example: you can add any DeskProto functionality here, which makes scripting a very powerful tool for automation.

```
var strSampleLocation = DPPreferences.getSampleLocation();

DPProject.loadGeometry(strSampleLocation + "Bottle.stl");
DPActivePart.setRotation(-90.0, 0.0, 0.0);
DPActivePart.segmentMethod = 2;
DPProject.calculateToolpaths();
DPProject.writeNCProgram( "ScriptOutputNCfile.ext" );
DPProgram.exit();
```

The second example script file that has been installed is called ***ScriptedBottle.dps***
This script will load the bottle sample geometry, rotated and segment it (Part Segment is the old name for the Material block, still used in the Scripts), calculate toolpaths and save an NC file. No path for the NC Program file is specified, so it will be saved in the default location for Write NC which is in most cases is the (My) Documents folder.

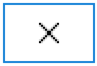
When using such script as command line parameter you can even add an Exit command: then the actual user does not need to perform any user-interaction with DeskProto.

For more information on scripting see the DeskProto Script Reference. You can find that on the DeskProto distribution CD, or without such CD email us to obtain a copy.

3.1.18 Exit

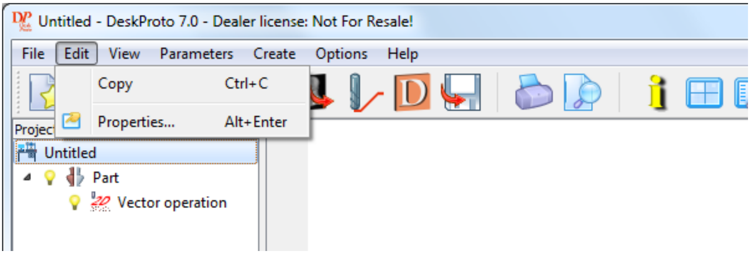
This File menu command ends your DeskProto session. DeskProto prompts you to save projects with unsaved changes.

Shortcuts:

Mouse: Click the application's Close button  (top right button of the title bar).

Keys: ALT+F4

3.2 Edit Menu



The Edit menu contains all options for Clipboard actions and a Properties command, conform Windows conventions. The number of commands is very limited though as DeskProto does not include an Undo function, and as Cut, Paste, Select and Search actions are not applicable for DeskProto.

3.2.1 Copy

This Edit menu command is applied to the currently active item in DeskProto.
That can either be one of the lines in the Project Tree or the image in the View Window.

In the Tree the Project cannot be copied, so when that line is active an error will pop up.
In case a Part or an Operation is active it will simply be copied: a copy of the Part or Operation will be added to the Tree.

The image in the DeskProto View Window will be copied to the Windows Clipboard.
The clipboard is used to cut and paste data between windows applications: you can for instance Paste this image into a Word document. Copying data to the clipboard replaces the contents previously stored there. This command has no visible effect in DeskProto.

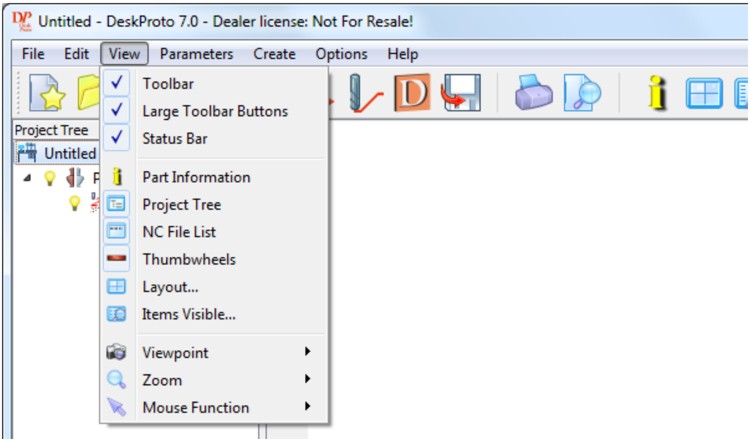
Shortcuts:
Keys: CTRL+C

3.2.2 Properties

This Edit menu command can be used to edit the highlighted tree-item or the active view. Depending on what is selected last this command will open one of the following five dialogs:
Project Parameters , Part Parameters , Geometry Operation Parameters , Vector Operation Parameters , Bitmap Operation Parameters , Viewpoint .
This is conform Microsoft's specifications for Windows: Edit -> Properties should open the Properties dialog for the selected item.

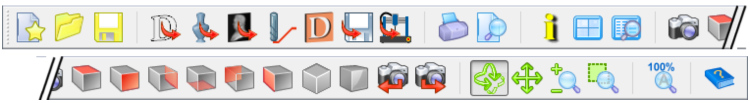
Shortcuts:
Keys: Alt+Enter

3.3 View Menu



The functions in the View menu let you control what is visible on your screen.

3.3.1 Toolbar

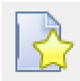


The toolbar is the series of buttons displayed across the top of the application window, below the menu bar. The toolbar provides quick access to many tools used in DeskProto. All these functions can also be accessed via the menus.

To hide or display the Toolbar, choose Toolbar from the View menu. A check mark appears next to this menu item when the Toolbar is displayed.

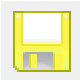
The next command in the View menu is "Large Toolbar Buttons", with which you can choose between two sizes of buttons. Again with a check mark if large buttons have been selected (which is the default status). The best choice will depend on the resolution of your screen: on a high-res screen the small buttons will become too small.

Below a list of all buttons with for each button an explanation of its functions.



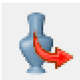
Open a New project. Same command as New in the File menu

Open an existing project. DeskProto displays the Open dialog, in which you can locate and open the desired DPJ-file.

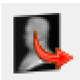


Save the open project with its current name as DPJ file. If you have not yet named the project, DeskProto displays the Save As dialog.

Load or Add a Vector file into the project.



Load or Add a Geometry file into the project.

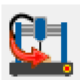


Load a Bitmap file into the project (adding not possible: max 1 bitmap file loaded).

Calculate the toolpaths for all visible operations of the current part.

Show a Simulation of the result that you can expect after machining the part.
This is a "toggle button": a button that can be switched on (depressed) and off (normal state), and show's its current state.

Write the NC program for all visible operations of the current part.

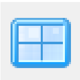


Send toolpaths to machine for all visible operations of the current part.
This button is visible ONLY when this option has been configured (in the Preferences); for some configurations the the button will show a different icon.

Print the image as present in the current view.

Preview how the image would be printed.

Show or hide the Part information dialog.
This is a "toggle button": a button that can be switched on (depressed) and off (normal state), and show's its current state.



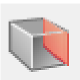
Change the Layout out of the views.

Change which Items should be shown in the active view (the Scene).

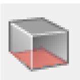
Change the Viewpoint (camera position) of the active view.

Set the viewpoint of the active view to Top view (XYZ 0 / 0 / 0).

Set the viewpoint of the active view to Front view (-90 / 0 / 0).



Set the viewpoint of the active view to Right side view (-90 / -90 / 0).



Set the viewpoint of the active view to Bottom view (0 / 180 / 0).

Set the viewpoint of the active view to Back view (-90 / 180 / 0).

Set the viewpoint of the active view to Left side view (-90 / 90 / 0).

Set the viewpoint of the active view to Isometric view.

Set the viewpoint of the active view to Default view.

Restore the previous viewpoint settings.

Restore the next viewpoint settings (enabled only after Restore previous view, to undo that restore).

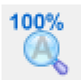
Change mouse-function to rotation.
The four mouse-buttons are "toggle buttons", showing depressed in case selected. De-selecting can be done by pressing one of the other mouse-buttons. Always one of these four buttons is selected.

Change mouse-function to panning.

Change mouse-function to zooming.

Change mouse-function to zoom window: zooming in by selecting a specific area in the active view.

Note that one of these four mouse-functions is active at any moment, so choosing one means deselecting the previous one. DeskProto also offers other tools: the red thumb-wheels on the screen are for rotations, the yellow for pan and the blue wheel is for zoom. A handy alternative is using the middle mouse button (the mouse-wheel): rotating this wheel zooms, and moving the mouse with the wheel pressed pans.



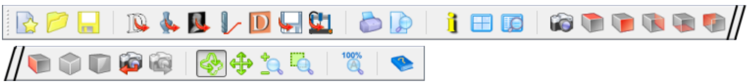
Set the Zoom percentage to 100% to show the complete geometry.

Display the Help Topics dialog.

3.3.2 Large Toolbar Buttons

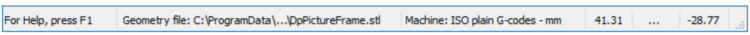
The Toolbar is the series of buttons displayed across the top of the application window, below the menu bar.
In case your have a high resolution monitor - or if you do not have clear eye-sight - these buttons might be too small for you to clearly recognize them. To solve that problem you can ask DeskProto to display large size buttons. Note that in the Windows Display properties you can select a larger font for all text.

To select or deselect this option, choose "Large toolbar buttons" from the View menu. A check mark appears next to the menu item when the large buttons are displayed.



The illustration shows the difference between large size and small size.

3.3.3 Status Bar



The status bar is displayed at the bottom of the DeskProto window, and gives status information about various relevant items. To display or hide the status bar, use the Status Bar command in the View menu. A check mark appears next to the menu item when the Status Bar is displayed.

The left side of the status bar describes the actions of toolbar buttons as you point to them (a sort of extra Help information). Menu item actions are also described when you use the arrow keys to navigate through the menus.

In the middle area the most important parameters of the selected tree-item are displayed: CAD filename and machine for the Project, dimensions for a Part, cutter and precision for an Operation.

At the right side of the status bar the **coordinates of the current mouse position** are displayed, however only when the geometry is displayed in one of the main views. These coordinate values shown are in “Translated” coordinates: the coordinates as used in the NC file. This is a very handy option that enables you to quickly check dimensions and positions on the screen.

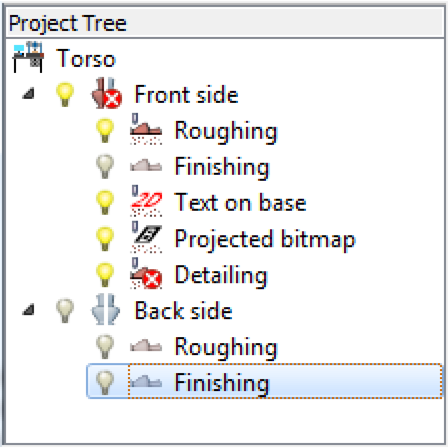
3.3.4 Part Information

The View menu command Geometry Information displays or hides the **Part Information dialog**, which shows information about the geometry and the current part. A check mark appears next to the menu item when the dialog is displayed.

Shortcuts

Toolbar: 

3.3.5 Project Tree



In DeskProto the **structure** of the Project is shown in the project tree, which is placed on the left of the screen. In the above example the project 'Torso' contains two Parts (icon), one with five Operations and one with two. Three different Operation types can be present: Vector operation (icon), Geometry Operation (icon), and Bitmap operation (icon). When one of these Operations or Parts is in an error status this is shown by the **error icon**.
The **lamp icon** indicates if a line in the Tree is visible (yellow) or not (gray). Only one Part can be current (visible) at a time. Of the Operations in that part none, several or all can be visible. Click on a lamp icon to change the status of that line. The Project line has no lamp icon as that line cannot be turned off.

If you do not see a Project tree window, activate the option Project tree in the View menu by selecting it. The Project tree icon in the menu will become active (drawn in a frame) and the Project Tree will be displayed. De-activating this option (by again selecting it) will make the Tree window disappear. The size of the Window can be changed by dragging it's borders with the mouse.

Shortcuts:

The black arrow button in the border of the View Window opens and closes this Tree Window.

The project tree offers you a number of functions:

Editing parameters
Double-clicking on a line in the tree will open the dialog to edit the parameters of the Project, Part or Operation.

Making a part current
To see a particular part (in case you have defined more than one part) you should make it current. To make a part current just click with the left mouse-button on the gray lamp icon of that part. You can also use the Context menu (see below). When a part is not current, its icon is grayed.
There is always exactly one part that's current. No more, no less.
You cannot make a Part "un-current" by clicking on a yellow lamp icon: you can only make a different part current.

Making an operation (in)visible
To be able to see how the toolpaths look for a particular operation, this operation should be visible. To make an operation visible just click with the left mouse-button in the tree on the gray lamp icon of that operation to make it yellow (turn the light on). Clicking on the yellow lamp icon of a visible project will make it invisible. When an operation is not visible, its icon is grayed.
Of the operations in a Part none, one, several or all can be visible.

Displaying parameters in the status bar
When you single-click on one of the items in the project-tree, that particular item is highlighted (shown with a blue background), meaning that it is **selected**. At that moment some of the parameters of that item will be shown in the Status bar.

Context menus
When you **right-click** on an item in the tree a small menu pops up called the context menu, offering you a number of functions (right-clicking on a Mac with a one-button mouse can be emulated by pressing the Ctrl (Control) key while clicking the mouse button).
The available functions will be different for each line of the tree, and will include the following options:

Edit the parameters of that tree-item.

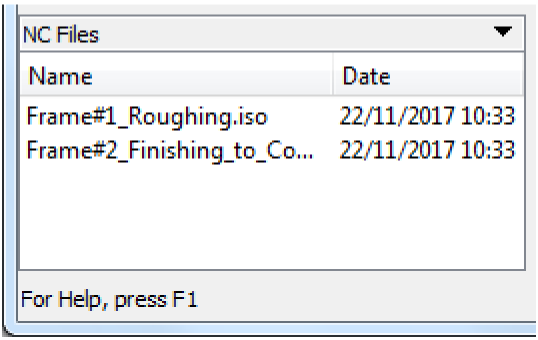
Add a Part to the project. The settings of the default part are used.
Add an Operation to that part. The settings of the first default operation are used. There are Add options for a Vector Operation, a Geometry Operation and a Bitmap Operation.
Copy an operation or a part will add an item that is identical to the current one.

Remove a Part from the project. This is only possible when there will be at least one part left after it has been removed.
Remove an Operation from that part. This is only possible when there will be at least one operation left in the part where it belongs to after it has been removed.

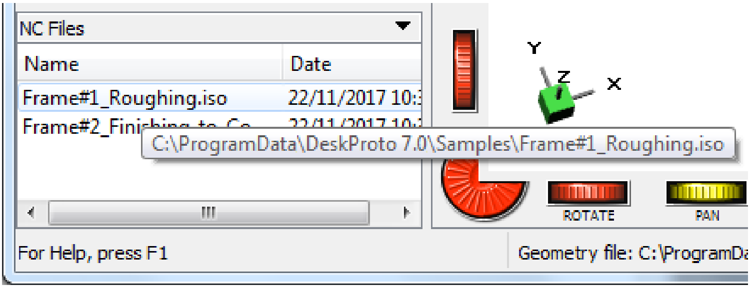
Move Parts and Operations in the Tree can be used when the sequence is important (for instance first roughing, then finishing).

The other options in the context menu will be self explaining.

3.3.6 NC File List



The NC files window shows a list of NC program files that have been saved for this project. For each file its Name and Date are listed. This may be easy for you to manage NC files for this project.



Hoovering over the filename (positioning the cursor there without clicking) will make DeskProto show the complete file specification of the NC file in a tooltip: see the screenshot above.

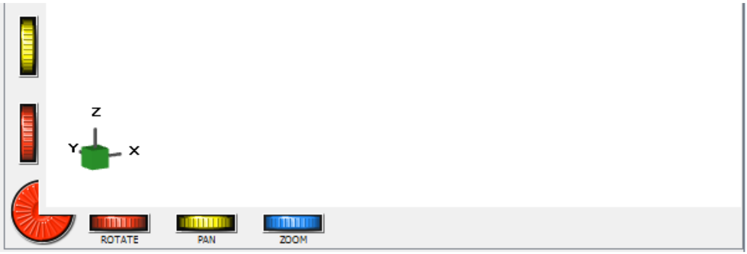
Double-clicking a file will open the file using the default program that has been set for that file-type in File Explorer. For instance double-clicking a .TXT file will open it in Notepad. You can for instance configure your machine's control software to open your NC files, or a text editor in case you want to check the file's contents.

By right-clicking on any line in this window you can open a Context menu, offering the following options:

- Remove the file-name from the list
- Delete the file
- Open the file - same effect as the double-click just described
- Open the file location - which will show the NC program in File Explorer
- Send the file to your machine (only in case that option has been configured)
- Add a file - for instance to add a TXT file with your own project documentation.

The NC Files window can made visible or invisible by checking or unchecking the option NC Files list in the View menu. This same effect can be achieved by pressing the black arrow button in the title bar of this window. Note that the NC Files window can only be visible when the Project Tree Window is visible too.

3.3.7 Thumbwheels



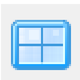
The **Thumb-wheels** that are drawn in the border of the View window offer an easy way to change the camera position. You can use them by pressing the left mouse button with the cursor on the wheel and then moving the mouse, keeping the left button pressed. The cursor will become arrow-shaped to guide you. You can look at the small green axis cube (the **Orientator**) in the left-bottom corner of the view to help you when rotating. The three red thumb-wheels control the rotation (three axes), the two yellow wheels control the pan (horizontal and vertical movement), and the blue one controls the zoom.
The Thumb-wheels can be switched on and off in the View menu.

Note that these rotations only change the viewing angle (camera position), not the orientation of the part in space.
You can make the thumb-wheels visible and invisible using the Thumb-wheels command in the View menu.

3.3.8 Layout

The View menu command Layout... displays the Views Layout dialog in which you can change the layout of the views in which the geometry is drawn. You can view either 1, 2, 3 or 4 views at the same time.

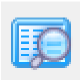
Shortcuts:

Toolbar: 

3.3.9 Items Visible

The View menu command Items Visible... displays the [Items Visible dialog](), in which you can change what is and what is not shown in the active view (the Scene). In case none of the boxes is checked the View Window will be blank. You can also select which operations should be visible.

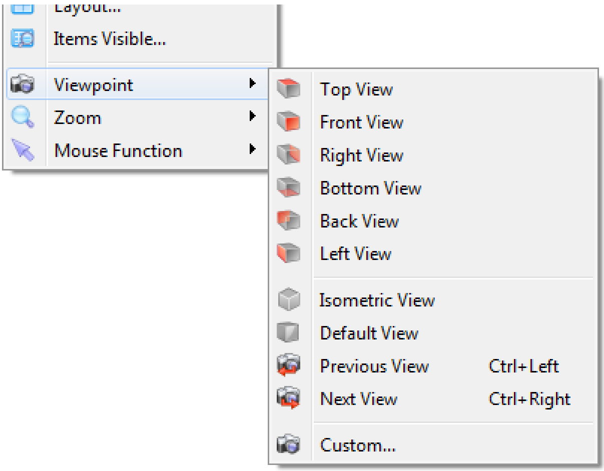
Shortcuts

Toolbar: 

Mouse: Right-click in a View, and select Items Visible in the shown context-menu.

It is even quicker to just double-click inside a View.

3.3.10 Viewpoint



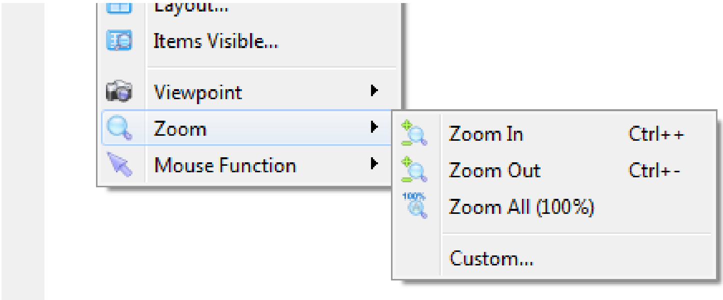
The View -> Viewpoint submenu offers the following commands to set the Viewpoint:

Top / Front / Right / Bottom / Back / Left to set one of the six main views.
Isometric / Use default to set the Isometric or the [Default view]().
Previous to return to the previous View settings.

Next (enabled only after Restore previous view) to undo that restore.
Custom will open the Viewpoint dialog.

The same functions can be accessed more easily using the Toolbar buttons.
Note that the Viewpoint can also be set using the Thumb-wheels or the Mouse functions.

3.3.11 Zoom

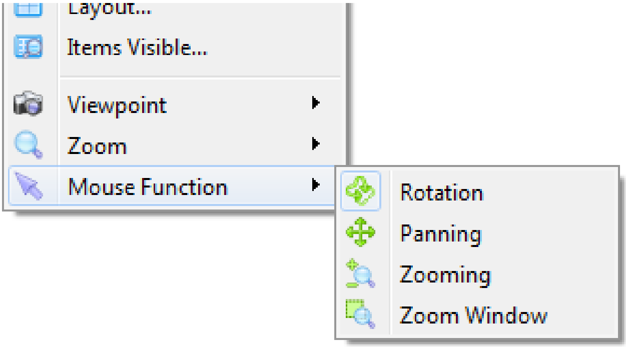


The View -> Zoom submenu offers the following commands:

Zoom In and Zoom out both change the zooming factor with 15 %.
Zoom All sets the zooming factor to 100 % to completely show all items.
Custom... will open the Viewpoint dialog, as the zooming factor is one of the Viewpoint settings.

The latter two functions can be accessed more easily using the Toolbar buttons.
Note that the Zoom can also be set using the Thumb-wheels or the Mouse functions.

3.3.12 Mouse Function



This command determines the functionality offered by the left mouse button inside the View Window. Four different functions are possible, of which always exactly one is active (the four functions are toggled). In the Toolbar you can quickly see which function is active as that button is drawn depressed.

Rotation: use the mouse (move it inside the graphics view with the left mouse button pressed) to rotate your geometry. Imagine that the geometry is inside a large hollow glass sphere: with your mouse you can grab the sphere anywhere, and rotate it round its center point, including the geometry. This means that grabbing and moving (say) left in the upper part of the screen has a different result than grabbing and moving left in the lower part of the screen. Note that in fact the geometry is not rotated, but the camera position (viewpoint) instead. You can see this, as during rotation the Orientator (the small axis cube at the bottom left of your screen) rotates with the geometry. If you want to rotate the geometry you should use the rotation option in the Part Parameters.

Pan: use the mouse to pan your geometry (move it on the screen, left-right, up-down, etc). When zoomed in you can use panning to determine which part of the geometry to look at.

Zoom: use the mouse to zoom in and out: move the mouse up is zoom out (push away), move down is zoom in (pull in). The center of the screen remains directed to the same position.

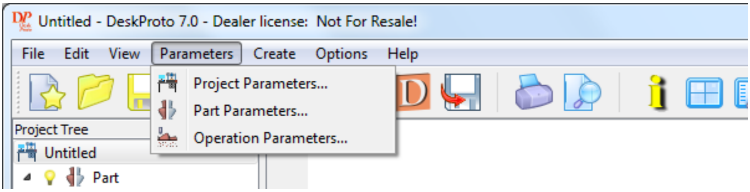
Zoom Window: use the mouse to zoom into any part of the screen. Click the left mouse button to define one corner of a bounding box, move the mouse keeping the button pressed, and release it as you have reached the opposite corner. The part of the screen inside the bounding box will now be displayed as large as possible.

Using the first three functions the geometry is continuously redrawn during the mouse movement. Depending on the size of your file and the speed of both your computer and graphics card, this redrawing will take more or less time. In case the redrawing is too slow, you can influence the number of entities to be continuously redrawn: see Options - Preferences - Advanced.

An alternative for *Zoom* is rotating the **Mouse wheel**. This works no matter which of the above buttons is active. In the Preferences you can reverse the effect of the rotation direction.
An alternative for *Pan* is moving the mouse with the middle button (the wheel) depressed.
So when the mouse-function button “Rotation” on the screen is active for the left mouse button, using these alternatives you have Rotate, Pan as well as Zoom easily available without having to press any Toolbar button.

Note that you can also use the red, yellow and blue Thumb-wheels on the border of the graphics screen.

3.4 Parameters Menu



The parameters menu offers access to all milling parameters. The three main levels Project, Part and Operation follow the structure of the Project Tree: one Project can contain one or more Parts (for instance a left half and a right half), and each Part can contain one or more Operations (for instance for roughing, finishing and detailing). Operations can be one of three types: Vector Operation, Geometry Operation and Bitmap Operation.

3.4.1 Project Parameters

This Parameters menu command displays the Project parameters in which you can edit the parameters of your current Project.

This dialog can be reached via the Parameters menu, first option.

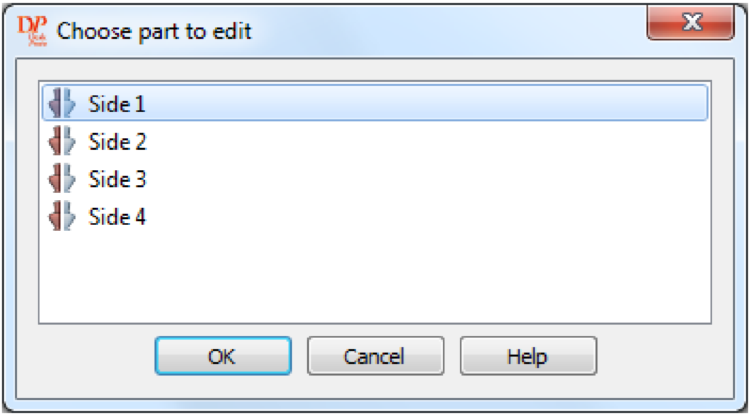
Shortcuts:
You can double-click on the project-item in the project tree (the base level item).
Or right-click on the project-item and select Project Parameters in the context-menu.

This same dialog is used for the Default Project parameters.

3.4.2 Part Parameters

This Parameters menu command displays the Part Parameters dialog in which you can edit the parameters of a Part.

In case a Project has more than one Part, first a dialog will be shown in which you can select the part you want to edit.



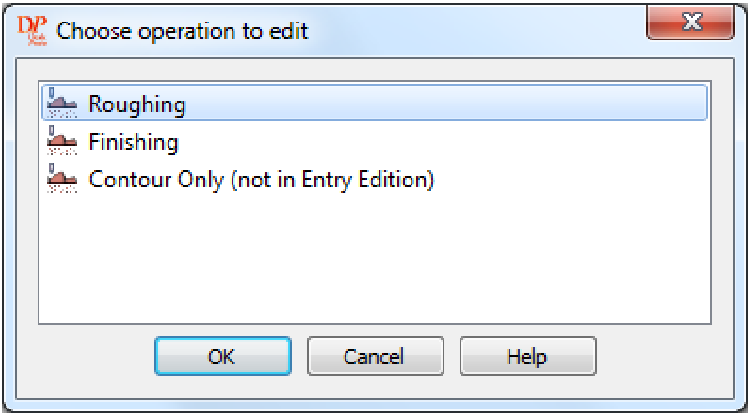
The part you select here will also become the current Part, and thus will be displayed when you have finished editing.

Shortcuts
Double-click on a part-item in the project tree (one of the second level items). Or right-click on a part-item and select Part Parameters in the context-menu.

3.4.3 Operation Parameters

This Parameters menu command displays the Operation Parameters dialog in which you can edit the parameters of an Operation.
Operations can be either Geometry Operations , Vector Operations or Bitmap Operations.

In case a Part has more than one Operation, first a dialog will be shown in which you can select the operation you want to edit.

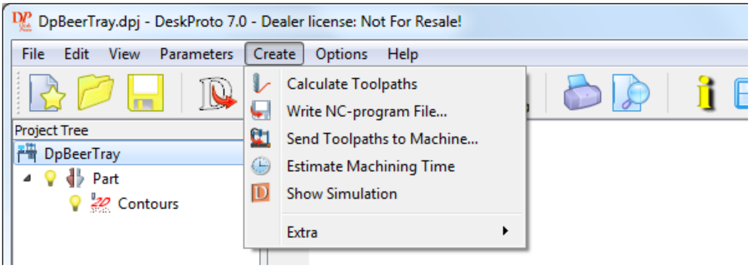


Shortcuts

Double-click on an operation-item in the project tree (one of the third level items).

Or right-click on a operation-item and select Operation Parameters in the context-menu.

3.5 Create Menu

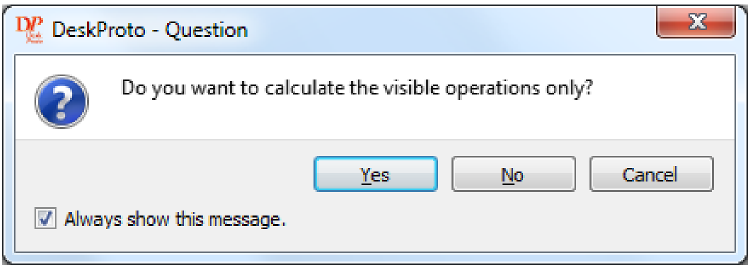


These options control all actions for calculating and saving NC toolpaths. Note the submenu called Extra which offers some extra options. As these are very specific and not important for most users these commands have been "hidden" in a submenu.

3.5.1 Calculate Toolpaths

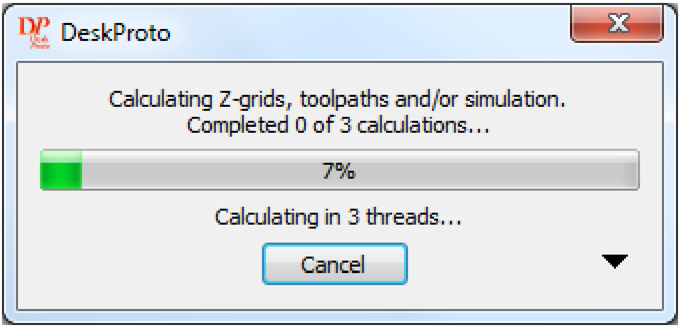
With this option you can create the toolpaths for the current part.

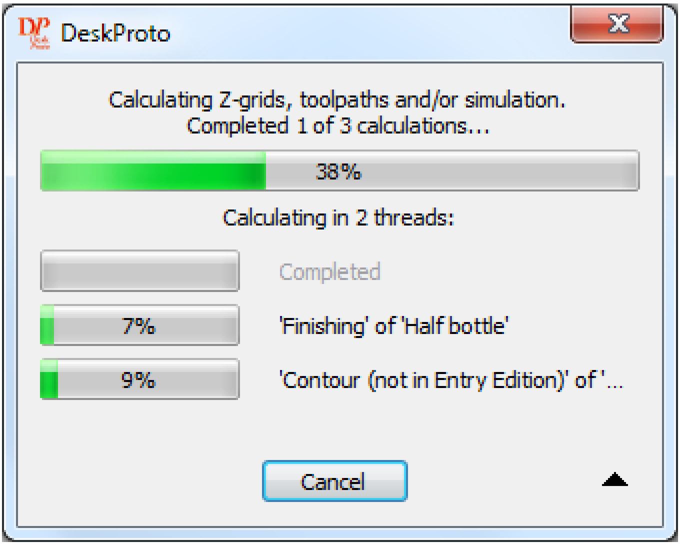
In case all operations of the current part are visible (see Visible Operations), all the toolpaths for the current part will be calculated and drawn. When at least one operation of the current part is not visible, you will be asked if you only want to use only the visible operations for calculations or if you want to use all operations.



In case you choose No for ‘all operations’, they all will be made visible after calculations.

It may be that you grow tired of this warning popping up all the time. Then you can disable "**Always show this message**" by removing the mark from the checkbox: the warning then will no longer be displayed. You can reset it on the Advanced tab of the Preferences.





While calculating, DeskProto will show a **progress bar** like shown above. You can select either the small dialog left or the detailed view right using the black arrow (triangle) button in the lower right corner. DeskProto is a **multi-threaded** application, so it can split up the calculations over multiple cores (in case you have a multi-core PC).

Each operation is assigned it's own core (so when you project has just one operation the calculation will not be multi-threaded). Of course no more threads can be running at the same time than the number of cores that is available. The detailed view of the progress bar will show maximum 8 threads (even on a 16 core PC) as otherwise the dialog would become too large.

In case you want to calculate all toolpaths for ALL Parts, then you can use the command Calculate All Toolpaths in submenu Extra.

Shortcut:

Toolbar: 

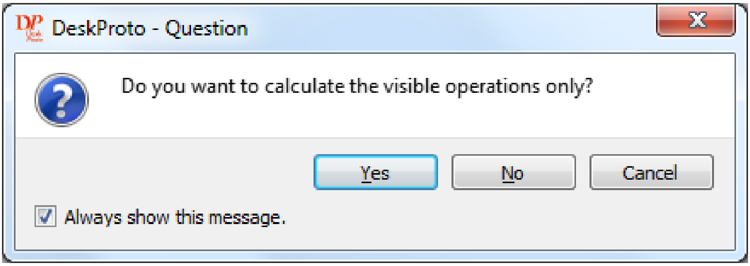
3.5.2 Write NC-program File

With this option you can create an NC-program-file for the current part, which can be sent to the machine to mill a part.

First you will be asked to give a **Name** for the NC-program file, in a standard Windows Save-As dialog, in which the correct file-extension for your machine has been already entered. The only thing you have to do are make sure the file is being saved in the right place and choose a filename.

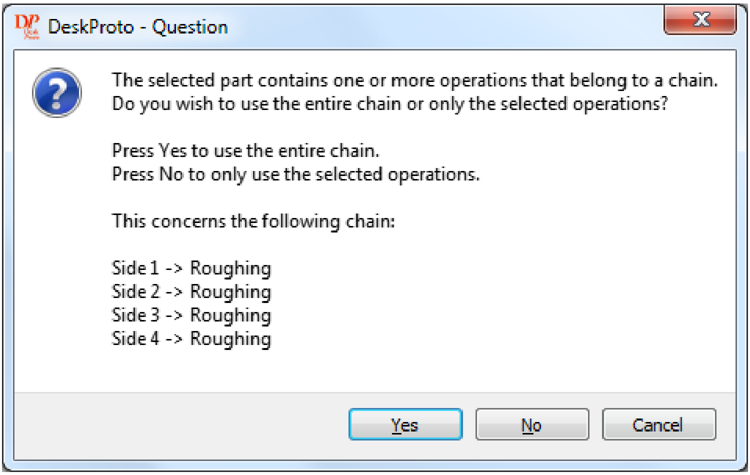
The name for the NC file that you enter here may be automatically changed. When you have more than one Operation and different cutters are used in these Operations, a tool change will be needed. In case your postprocessor specifies to start a new NC file at each tool change, two or more NC files will be written. Names for these subsequent files will be automatically generated. In case you chose the name Test.nc and the two operations were called Roughing and Finishing, then the first file will be called Test#1_Roughing.nc and the second file Test#2_Finishing.nc

In case all operations of the current part are visible (see Visible Operations), all the toolpaths of the current part will be calculated (in case they not already had been). When at least one operation of the current part is not visible, you are asked if you only want to use the visible operations for calculations or if you want to use all operations.



In case you choose No (so choose 'all operations'), they will also all be made visible after calculations.

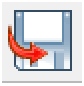
A next message box will pop-up in case one of the Operations is part of a Chain:



Normally the answer will be Yes, meaning that you want to write the complete toolpath, including all Operations that belong to this chain of Operations.

After that, the NC-program(s) will be created, using the post-processor that is configured for the machine you have selected in the current part.

Shortcut:

Toolbar: 

3.5.3 Send Current Toolpaths to Machine

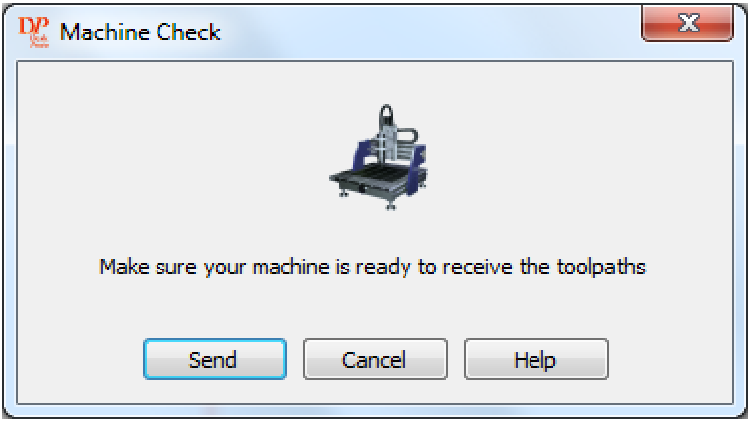
For many CNC milling machines this option can be used to directly send the toolpaths to the machine (or to the machine's control software) from DeskProto. As this is not possible on all machines, by default this option is not available (disabled: 'grayed out'). The option will become available after configuring an output destination to be used, in the DeskProto Preferences.

In the Preferences (Options menu) the following destinations can be configured for this option:

- a Windows Printer driver (this is possible for a few machines only, like the machines made by Roland)
- a port like COM1: or LPT1:
- an external program
- none, which is the default.

DeskProto will write a temporary NC program file (TempNC.ext, conform the machine that you selected and its postprocessor) and will open the external program with this file TempNC.ext as command line parameter. For the other options DeskProto will simply write all information to the selected port or driver (instead of to an NC file).

As external program you can configure the control software of your CNC milling machine. However in fact you can select any program, also for instance a milling simulation software, or (for the diehards) a plain text editor like Notepad to change the NC-program that DeskProto has just created.



After giving this command, the Machine Check dialog (shown above) will be displayed to ask you if the machine is ready. After pressing Send DeskProto will start sending.
In order to send an NC file you can use the command Send NC Program File to machine.

If your machine does not support this: the standard route to get the NC program file to the machine is to exit DeskProto and transfer the NC file using the machine's own communication software. If this software runs on a different computer you will first have to transfer the file to that PC, via a network or by USB stick or CD.

How to prepare the machine will be different for every machine. Still some general guidelines can be given. Basically the following steps have to be taken:

1. Fixture a fitting block of material on the machine (look in the Geometry Information dialog for the correct dimensions).
2. Mount the correct tool. Note: in case you use a different cutter than entered in DeskProto an incorrect part will be produced.
3. Tell the machine where to find the material. In other words: set the workpiece zero-point. By default DeskProto uses the left-front-top corner of the material block as zero-point, you can change this at Translation tab of the Part Parameters.
4. Send the NC-program file to the controller and start machining.

3.5.4 Estimate Machining Time

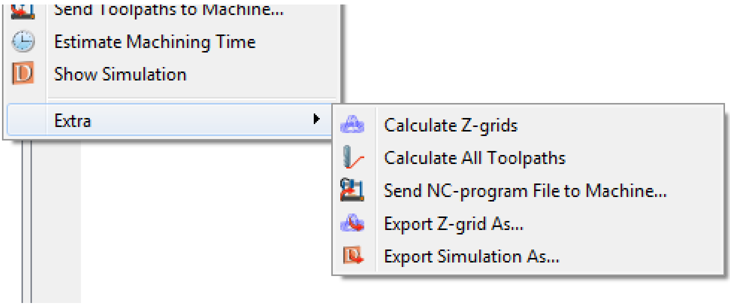
This command calls the Estimated Machining time dialog, in which DeskProto gives a rough estimation for the machining time that it will take to create your part. Be aware that this is a rough estimation: the actual time will be influenced by many factors.

3.5.5 Show Simulation

With this option you can create a Simulation for the current part.
The command will immediately show the block of material, so before any operation has been machined.
In the Simulated operations dialog that pop up you can select which operations you want to be processed and press Calculate to start.

This is a toggled command: again giving the same command will make the Simulation invisible.

3.5.6 Extra

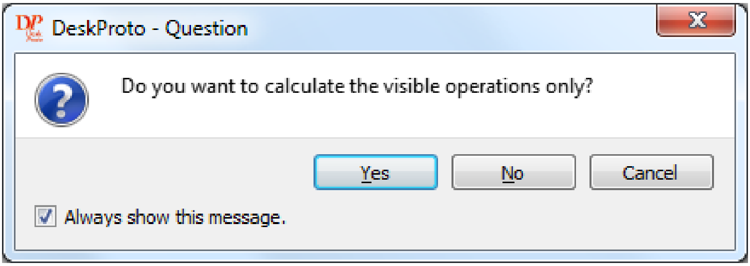


These Extra commands also are options for calculating and saving NC toolpaths. As these Extra commands are very specific and not important for most users these commands have been "hidden" in a submenu.

3.5.6.1 Calculate Z-grids

With this option you can create the Z-grids (an intermediate calculation result) for the current part.

In case all operations of the current part are visible (see Visible Operations) in the active view, all the Z-grids for the current part will be calculated. When at least one operation of the current part is not visible you are asked if you only want to use only the visible operations for calculations or if you want to use all operations.



In case you choose No for 'all operations', they all will be made visible after calculations.

3.5.6.2 Calculate All Toolpaths

With this option you can create all Toolpaths for all Parts.

Where the other commands in the Create menu only concern the current part, this one command influences all parts in the current project.

All operations for all parts will be calculated, so also the invisible operations (see Visible Operations).

3.5.6.3 Send NC-program File to Machine

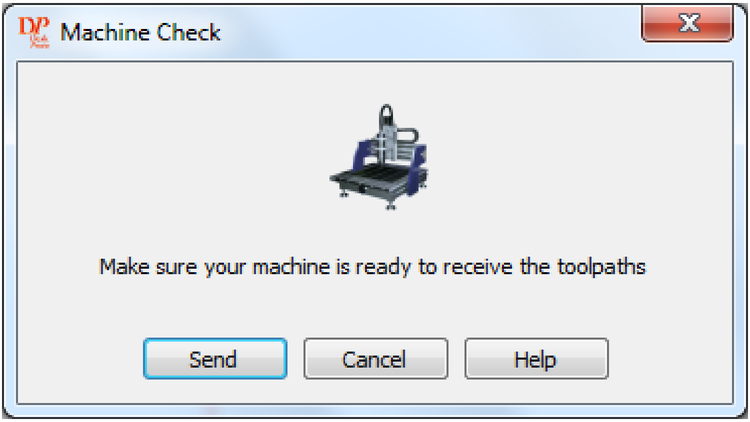
For many CNC milling machines this option can be used to directly send an NC program to the machine (or to the machine's control software) from DeskProto. As this is not possible on all machines, by default this option is not available (disabled: 'grayed out'). The option will become available after configuring an output destination to be used, in the DeskProto Preferences.

In the Preferences (Options menu) the following destinations can be configured for this option:

- a Windows Printer driver (this is possible for a few machines only, like the machines made by Roland)
- a port like COM1: or LPT1:
- an external program
- none, which is the default.

DeskProto will open the NC program file, and will simply copy its contents to the selected port or driver, or will open the external program with this file as command line parameter.

As external program you can configure the control software of your CNC milling machine. However in fact you can select any program, also for instance a milling simulation software, or (for the diehards) a plain text editor like Notepad to change the NC-program that DeskProto has just created.



In DeskProto the NC program file must have been saved first, in the correct format for your machine. A standard File Open dialog will be displayed to ask you for the file you want to transfer. After selecting the correct NC program file the Machine Check dialog (shown above) will pop up: after pressing Send DeskProto will start sending.
In order to send the current toolpaths without first saving an NC file you can use the command Send current toolpaths to machine.

If your machine does not support this: the standard route to get the NC program file to the machine is to exit DeskProto and transfer the NC file using the machine's own communication software. If this software runs on a different computer you will first have to transfer the file to that PC, via a network or by USB stick or CD.

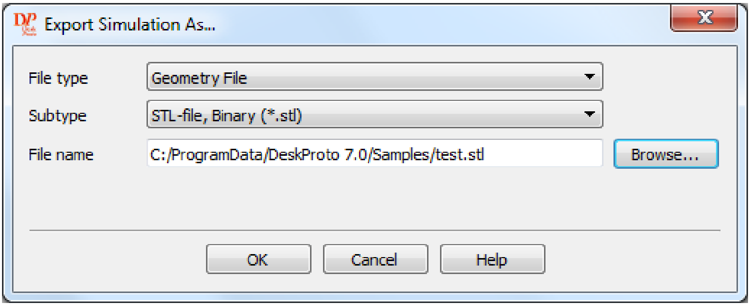
How to prepare the machine will be different for every machine. Still some general guidelines can be given. Basically the following steps have to be taken:

1. Fixture a fitting block of material on the machine (look in the Geometry Information dialog for the correct dimensions).
2. Mount the correct tool. Note: in case you use a different cutter than defined in DeskProto an incorrect part will be produced.
3. Tell the machine where to find the material. In other words: set the workpiece zero-point. By default DeskProto uses the left-front-top corner of the material block as zero-point, you can change this at Translation tab of the Part Parameters.
4. Send the NC-program file to the controller and start machining.

3.5.6.4 Export Z-grid / Simulation as

The Z-grid is a temporary representation of the geometry, used by DeskProto to calculate the toolpaths.
The Simulation has of course a completely different aim, still the internal representation in DeskProto is the same as for the rendered Z-grid. Both items are represented by a large number of triangle on the outer surface: polygon data.

As for geometry files in DeskProto the same polygon data representation is used, it is possible to export the Z-grid and/or the Simulation as a geometry file. For standard DeskProto use this is absolutely not needed, however it might be useful for instance to use external software to compare original geometry and simulation.



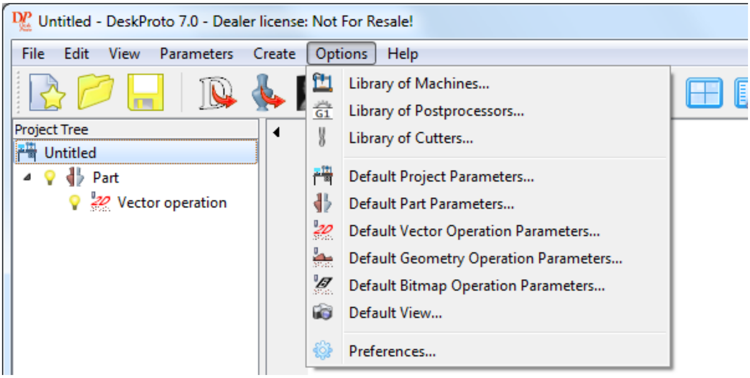
In the dialog box shown above you can choose how to export the Z-grid. For exporting the Simulation exactly the same dialog is used. The following options can be set:

The **File type** is either Geometry file, Bitmap file or XYZ file.
The use of a geometry file was described above.
For export as Bitmap file DeskProto will convert the 3D information to 2D by translating Z-height to gray-value. For each point in the Z-grid one pixel will be made. The highest Z-value will be given a white pixel, the lowest level a black pixel, and all in-between values an according in-between gray value. This is in fact the reverse procedure as used for the Bitmap Operation.
An XYZ file is a point cloud file: a number of points in 3D space, each point represented by its 3 coordinates.

The **Subtype** sets the file format that has to be used.
For Bitmap files you can choose one of four well known bitmap file types: BMP. GIF, JPG, PNG and TIFF.
For Geometry files you can choose any of the formats that DeskProto supports for 3D, as described in 3D Geometry.
For XYZ files only one format is supported: ASCII text file, one point with three coordinates per line.

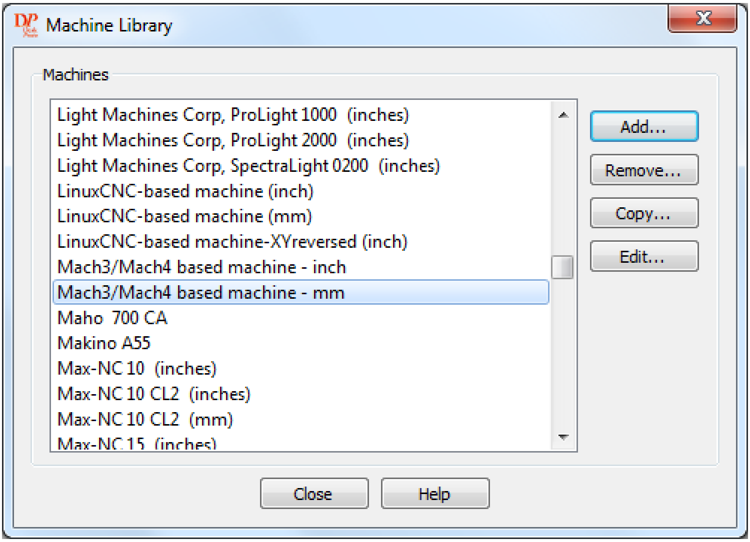
The function of **File name** will be clear, you need to use the Browse to fill this field or edit it's contents.

3.6 Options Menu



The Options menu gives access to all configuration options of DeskProto. For normal use you will not need these options, they are present in case the defaults set by the installation program should not match your wishes. The option used most will be the creation of a new cutter definition, as in many cases the default cutters will not match your real cutters.

3.6.1 Library of Machines



For every NC program to be created DeskProto will apply a machine-definition, which should of course be correct for the NC milling machine that you are going to use. A large number of predefined machine-definitions have been included with DeskProto. This is the Library of machines, which has been copied to your computer during Setup.

In most cases when you first start DeskProto you can simply select one of the existing machines in DeskProto's Welcome screen (only visible the first time that your start DeskProto). After that you still can change this setting in the (default) Project parameters.
However, in case you have a special machine, not present in the list, you can edit an existing machine definition or define your own machine in the Machine library.

You can find the Library of machines in the Options menu. When you call this library a warning message will be displayed that this option is meant for advanced users only. The warning is optional: when you remove the mark from checkbox "Always show this message" it will no longer be given. You can restore all warning messages on the Advanced tab of the Preferences (in group Settings).

After this warning the dialog shown above will pop up. Here you can choose the machine you want to **Edit** or **Copy**, and also **Add** and **Remove** machines. After pressing Add, Copy or Edit the Machine dialog will be shown, containing all parameters to define a machine in DeskProto.

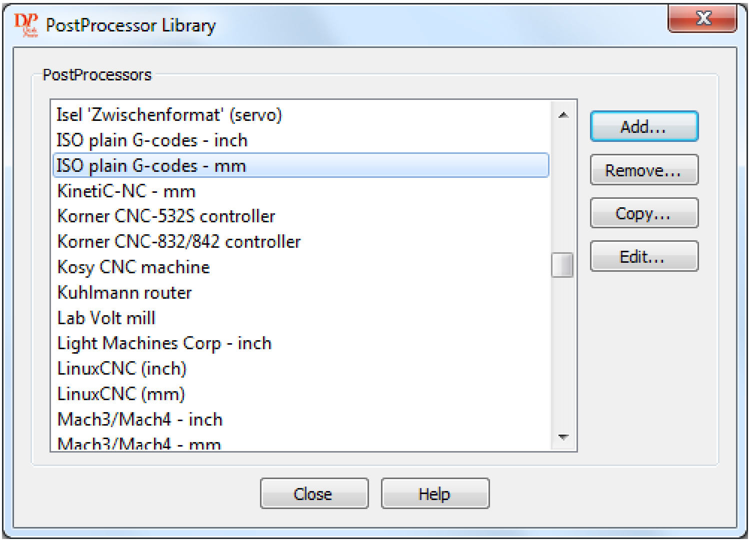
Before you start defining a new machine, be sure a postprocessor for the machine is already available. If not, configure one first with the function Library of Postprocessors (see the next paragraph).

A machine definition is stored as a file name .mch in the DeskProto drivers folder (See Preferences). These files can be copied, for instance to a different PC to make the machine available on that PC as well. You can easily find each driver file via command "**Open file location**" in the context menu: right click on any driver name in the list for this menu.

The files are in Windows .ini format, and can be changed using a plain editor like Notepad (changing in DeskProto is safer though). When changing in an editor, it is possible to add comments, for instance on the file history. Any line that starts with a semicolon (;) is a comment.

Note:
Selecting a machine here does NOT change which machine is selected for the current part. To select a machine for your current part open the Part Parameters dialog.

3.6.2 Library of Postprocessors



Every NC program created by DeskProto is made using a postprocessor. This is the part of the DeskProto software that is machine-dependent: it creates an NC program file that is exactly in the format required by your NC milling machine. In Windows terminology this piece of software should be called the device driver for a particular output device, however, in milling terminology it is called a Postprocessor and we will use that name. DeskProto makes it possible to define your own postprocessor (which is not possible for Windows drivers !). CNC machinists call this a ' Configurable Postprocessor'.

Note that you can not explicitly select the postprocessor that you want to use as one of the milling parameters: it will be implicitly selected when you select the milling machine. Each milling machine's definition has information about the postprocessor to be used.

You can find the Library of postprocessors in the Options menu. When you call this library a warning message will be displayed that this option is meant for advanced users only. The warning is optional: when you remove the mark from checkbox "Always show this message" it will no longer be given. You can restore all warning messages on the Advanced tab of the Preferences (in group Settings).

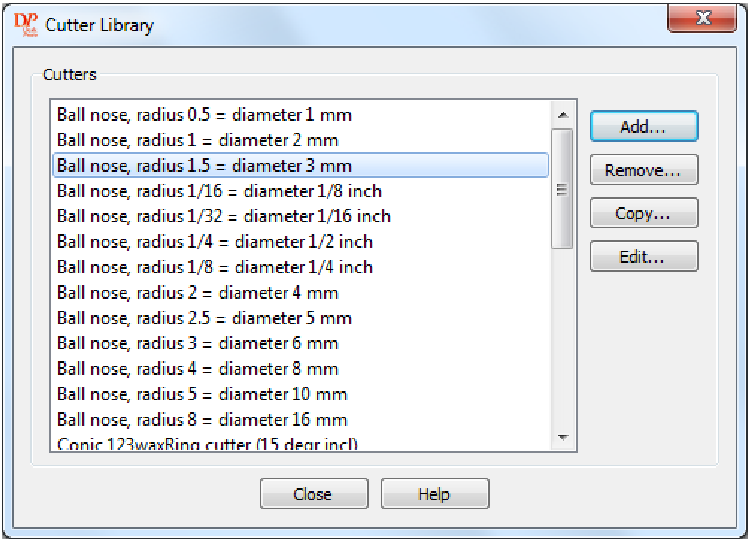
After this warning the dialog shown above will pop up. In this Library you can choose the postprocessor you want to **Edit** or **Copy**, and also **Add** and **Remove** a postprocessor. After pressing Add, Copy or Edit the Postprocessor dialog will be shown, containing all parameters to define a postprocessor in DeskProto.
Note: *as many parameters must be entered for a postprocessor definition, we recommend not to use Add to create a new postprocessor, but to Copy one that resembles the new one, and then Edit any changes needed. In most cases the postprocessor "ISO plain G-codes" is a good one to use as a start. Make sure to save it using a proper new name.*

A postprocessor definition is stored as a file name .ppr in the DeskProto drivers folder (see Preferences). These files can be copied, for instance to a different PC to make the postprocessor available on that PC as well. You can easily find each driver file via command "**Open file location**" in the context menu: right click on any driver name in the list for this menu.

The files are in Windows .ini format, and can be changed using a plain editor like Notepad (changing in DeskProto is safer though). When changing in an editor, it is possible to add comments, for instance on the file history. Any line that starts with a semicolon (;) is a comment.

Note:
Selecting a postprocessor here does NOT change which postprocessor is used to make an NC-program of the current part. A postprocessor automatically will be selected by selecting a machine. To use a particular machine in your project open the Part Parameters dialog.

3.6.3 Library of Cutters



For every NC program to be created (more accurately for every operation) you will need to select a cutter. Obviously the cutter that you select for DeskProto's calculations must be available for the actual milling process. A number of predefined cutter-definitions have been included with DeskProto. This is the Library of cutters, which has been copied to your computer during Setup. In many cases you can just select one of the existing cutters when editing the operation-parameters. However in case you need a special cutter you can change an existing cutter or define your own with this function 'Cutter Library'.

You can find the Library of cutters in the Options menu. When you call this library a warning message will be displayed that this option is meant for advanced users only. The warning is optional: when you remove the mark from checkbox "Always show this message" it will no longer be given. You can restore all warning messages on the Advanced tab of the Preferences (in group Settings).

After this warning the dialog shown above will pop up. Here you can choose the cutting tool you want to **Edit** or **Copy**, and also **Add** and **Remove** cutters. After pressing Add, Copy or Edit the Cutter dialog will be shown, containing all parameters to define a cutter in DeskProto.

A cutter definition is stored as a file name .ctr in the DeskProto drivers folder (see Preferences). These files can be copied, for instance to a different PC to make the cutter available on that PC as well. You can easily find each driver file via command "**Open file location**" in the context menu: right click on any driver name in the list for this menu.

The files are in Windows .ini format, and can be changed using a plain editor like Notepad (changing in DeskProto is safer though). When changing in an editor, it is possible to add comments, for instance on the file history. Any line that starts with a semicolon (;) is a comment.

Note:
Selecting a cutter here does NOT change which cutter is selected for the operations. To select a cutter to machine with open the Operation Parameters dialog.

3.6.4 Default Project

After choosing this option, first you will be warned that whatever you change here will influence all new projects that you create later. The warning is optional: when you remove the mark from checkbox "Always show this message" it will no longer be given. You can restore all warning messages on the Advanced tab of the Preferences (in group Settings).

The dialog that pops up is equal to the Project Parameters dialog, it will thus not be explained here. The difference is that this function adjusts the default project settings, that will be used for every new Project that is created.

The default project is either a Vector project, a Geometry project or a Bitmap project. You cannot define this **project type** in this dialog: the default project type is defined by setting an Operation of the preferred type as first operation in the Default Part.

Most important setting in this dialog is the default **Machine**. You can select any machine in the list to be used as default; if needed you can add and/or edit machines in the Library of Machines.

The functionality offered in this dialog is very limited: it is not possible to define a default geometry, nor more than one part. Only the options Use Z-values, Preserve Direction (Vector), Skip backfaces and Flip normals

(Geometry) are allowed. The DeskProto default for these four checkboxes is off.

Creating two or more Parts as default is not possible as that would make things way too complicated (for instance which of the operations then would be the actual default one...).
A default project file is not possible as that would conflict with the default Part and the default Operations.
Note that for repeating jobs can also achieve much using Command line parameters, by Scripting, and by auto-loading a template project file which may have more Parts. For instance for automation of two sided machining.

Note:
The default project parameters are stored in the registry. Each user has his/her own default settings stored there.
As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button is supplied here to **Restore the DeskProto defaults** for all parameters.

3.6.5 Default Part

After choosing this option, first you will be warned that whatever you change here will influence all new parts that you create later. The warning is optional: when you remove the mark from checkbox "Always show this message" it will no longer be given. You can restore all warning messages on the Advanced tab of the Preferences (in group Settings).

The dialog that then pops up is equal to the Part Parameters dialog, it will thus not be explained here. The difference is that this function adjusts the default part settings, that will be used for every new Part. You can use this for instance when you want to use more than one operation for all your parts, always apply a specific transformation or translation method, etc, etc.

For the default part all three groups of settings can be accessed:
- Vector settings
- Geometry settings
- Bitmap settings

even though no CAD data of that type is present here.

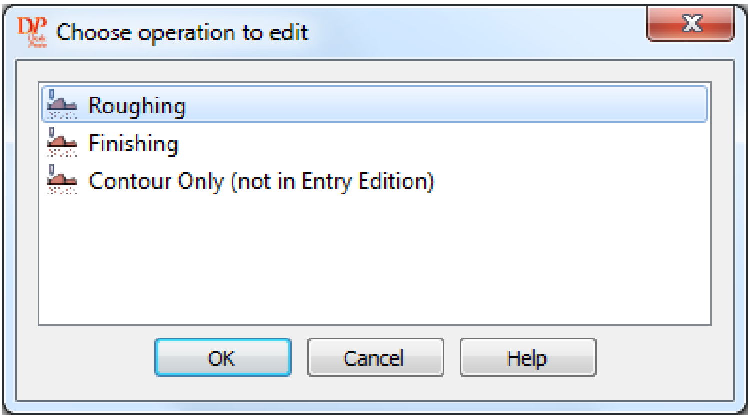
These default parameters will be stored in the Windows registry, and will be different for every user. As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button is supplied here to **Restore the DeskProto defaults** for all parameters. After using this button you will have to again choose your machine as the default machine.

3.6.6 Default Vector Operation

After choosing this option, first you will be warned that whatever you change here will influence all new Vector operations that you create later. The warning is optional: when you remove the mark from checkbox "Always show this message" it will no longer be given. You can restore all warning messages on the Advanced tab of the Preferences (in group Settings).

The dialog that then pops up is almost equal to the Vector Operation Parameters dialog, it will thus not be explained here. The difference is that this function adjusts the default Vector operation settings, that will be used for every new Vector Operation. This is the option to use for instance in case you want a specific cutter automatically selected, need a specific feedrate or spindle speed, etc, etc. These default parameters will be stored in the Windows registry, and will be different for every user.

In case more than one default Vector Operation has been defined in the default part, first a dialog is displayed in which you can select one of the default operations:



In this case the dialog will of course show Vector operations to choose from.

Two extra buttons are present, that are not available in the normal Vector Operation parameters:

As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button **Restore Defaults** is present here to Restore the DeskProto defaults for all parameters.
The DeskProto defaults are the built in values that were also present when you first installed DeskProto.
The default cutter might be different though: for a Vector operation DeskProto will select the first flat end cutter in the Cutter library with a diameter of 6 mm (metric users) or 1/4" (inch users). It such cutter is not present the first cutter in the library will be used.

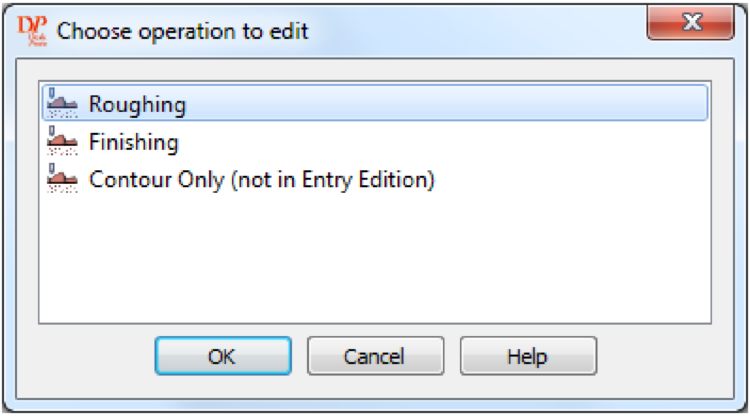
And on tab page Profiling and extra **Set...** button is present, in section Support tabs, to define the Default profiling support tab settings.

3.6.7 Default Geometry Operation

After choosing this option, first you will be warned that whatever you change here will influence all new operations that you create later. The warning is optional: when you remove the mark from checkbox "Always show this message" it will no longer be given. You can restore all warning messages on the Advanced tab of the Preferences (in group Settings).

The dialog that then pops up is equal to the Operation Parameters dialog, it will thus not be explained here. The difference is that this function adjusts the default operation settings, that will be used for every new Operation. This is the option to use for instance in case you want a specific cutter automatically selected, need a specific feedrate or spindle speed, etc, etc. These default parameters will be stored in the Windows registry, and will be different for every user.

In case more than one default operation has been defined in the default part, first a dialog is displayed in which you can select one of the default operations:



As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button is supplied here to **Restore the DeskProto defaults** for all parameters.

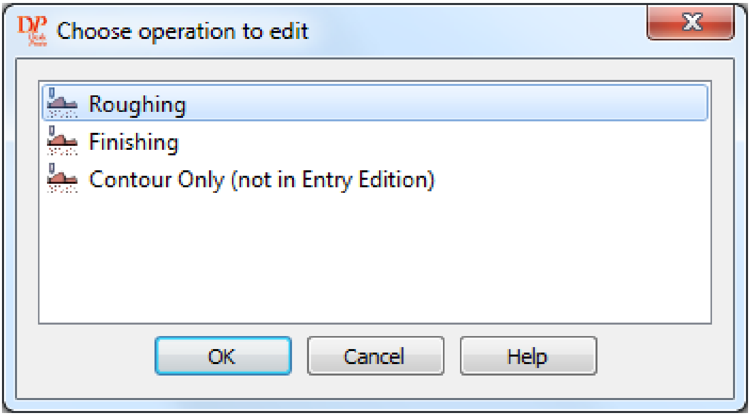
The DeskProto defaults are the built in values that were also present when you first installed DeskProto.
The default cutter might be different though: for a Geometry operation DeskProto will select the first ballnose cutter in the Cutter library with a diameter of 6 mm (metric users) or 1/4" (inch users). It such cutter is not present the first cutter in the library will be used.

3.6.8 Default Bitmap Operation

After choosing this option, first you will be warned that whatever you change here will influence all new Bitmap operations that you create later. The warning is optional: when you remove the mark from checkbox "Always show this message" it will no longer be given. You can restore all warning messages on the Advanced tab of the Preferences (in group Settings).

The dialog that then pops up is equal to the Bitmap Operation Parameters dialog, it will thus not be explained here. The difference is that this function adjusts the default bitmap operation parameters, that will be used for every new Bitmap Operation. This is the option to use for instance in case you want a specific cutter automatically selected, need a specific feedrate or spindle speed, etc, etc. These default parameters will be stored in the Windows registry, and will be different for every user.

In case more than one default Bitmap Operation has been defined in the default part, first a dialog is displayed in which you can select one of the default operations:

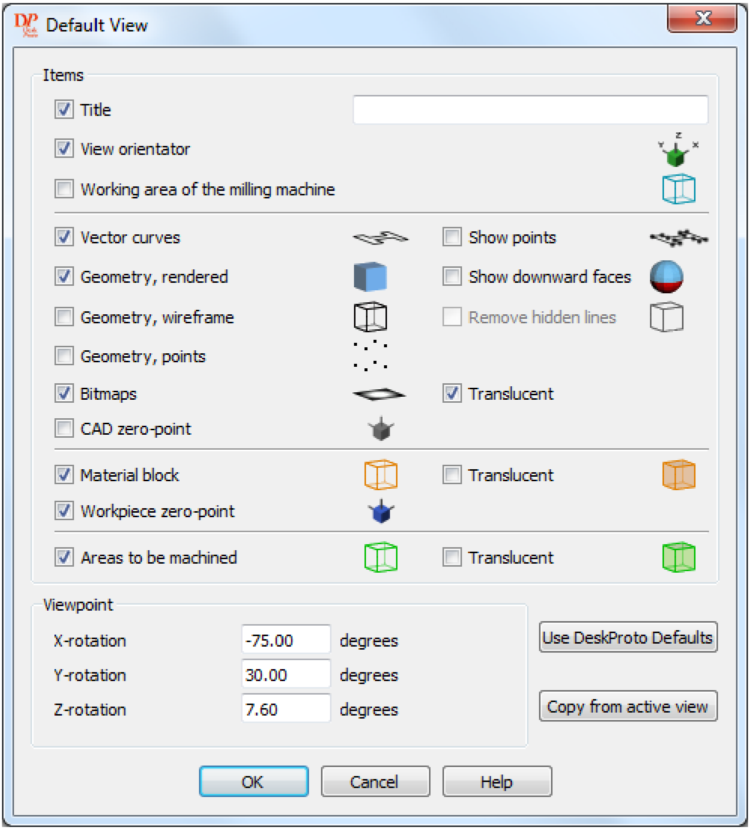


In this case the dialog will of course show Bitmap operations to choose from.

As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button is supplied here to **Restore the DeskProto defaults** for all parameters.

The DeskProto defaults are the built in values that were also present when you first installed DeskProto.
The default cutter might be different though: for a Bitmap operation DeskProto will select the first ballnose cutter in the Cutter library with a diameter of 6 mm (metric users) or 1/4" (inch users). It such cutter is not present the first cutter in the library will be used.

3.6.9 Default View



The default view settings are used for any new project and any new part. They are also used for the command and button Use Default View.

After choosing this option, first you will be warned that this option will only influence the defaults. The warning is optional: when you remove the mark from checkbox "Always show this message" it will no longer be given. You can restore all warning messages on the Advanced tab of the Preferences (in group Settings).

To edit the parameters of the default view a special dialog is used, as both the dialogs to change the Items and the Viewpoint contain parameters that are not suitable as default parameters.

Items
Here it’s possible to set what will be shown initially. For an explanation of the Items see Items visible dialog. Displaying Z-grids, toolpaths or simulations is not possible in the default view, as these items have to be calculated first.

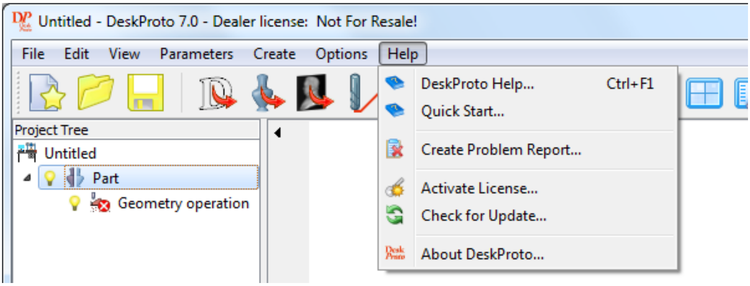
Viewpoint
Only the rotation can be set here. Zooming of the default view will always be 100%, panning will be set to zero.
After changing the default rotation you can restore the original **DeskProto defaults** (which are stored internally in DeskProto and which can not be changed) by pressing the appropriate button.
Finally: instead of typing three values for X, Y and Z rotation, you can manually adjust the rotation in your current view until you like it, and then copy the settings, by pressing the button **Copy from Active View**.

Note:
The default view settings are stored in the registry. Each user has his/her own default settings stored there.

3.6.10 Preferences

This is open the Preferences dialog, in which you can edit DeskProto's preferences. The dialog consists of 5 Tab pages.
These preferences are stored in the registry. Each user has its own preferences stored there.

3.7 Help Menu



Finally the Help menu which gives access to the Help system and to the About box.
For the DeskProto Editions Entry and Expert an extra option is present: Upgrading... (to a higher edition).

3.7.1 DeskProto Help

The **Contents** page of the DeskProto Help System, shown at the Contents tab on the left of the Help Window, offers an overview of all available Help topics, organized in 'books'. Each book can be opened by double-clicking its line or by clicking on the arrow icon in front of the line. Five books are available:

- Introducing DeskProto
- Screen Layout
- Menu commands
- Dialogs
- Concepts

The same five books can be found as Chapters in the DeskProto **Reference Manual** (available as PDF for download), which contains almost exactly the same information as the Help file.

In addition to the Contents tab page three more tabs are available:
the **Index** page can be used to find help on a specific subject
the **Bookmarks** tab to mark pages that you want to remember
the **Search** page to locate any word or phrase in the Help system.

While reading the Help text, additional related information can easily be accessed by clicking on the active items (Hyperlinks) in the text. Active words are shown in blue and underlined, while the cursor will turn to a hand when moved over an active item.

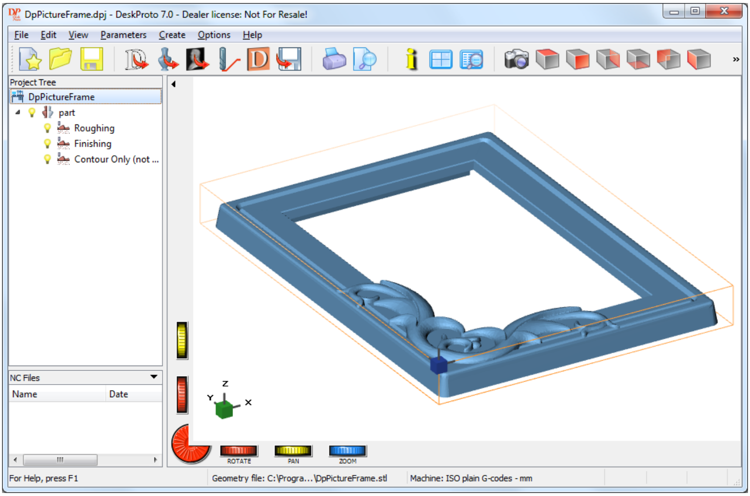
The online Help system that has been installed with DeskProto can be started from the Help menu. In addition some other methods to open Help are available as well:

- The Help button on the Toolbar: 
- Pressing the F1 Function key on the keyboard
- **All dialogs also have a Help button, providing Help information about the use of that particular dialog**.

3.7.2 Quick Start

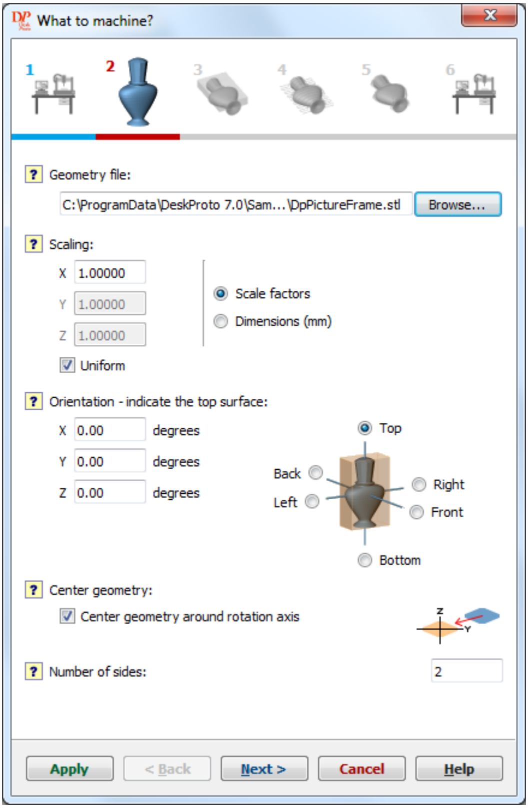
DeskProto is delivered with a Tutorial book (either printed or as PDF download), meant to step by step introduce you to the functions that DeskProto offers. It is recommended to read **and** execute at least lessons number one and two of this Tutorial before starting to make parts with your own CAD-data.

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. After that, novel users are advised to first use the Wizards, that will guide them through all steps needed to generate an NC program file.



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. Follow the links for more information on any item.

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

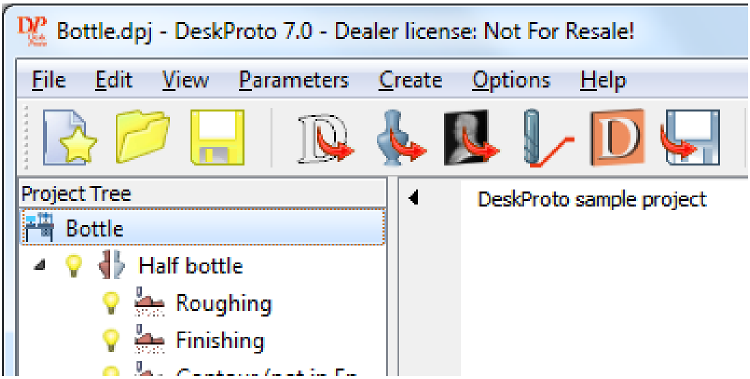


[1] Wizard-based interface.
New users are advised to use the DeskProto [Wizards](), that will guide them through all the steps needed to generate an NC toolpath file using their own CAD-data. The illustration above shows a typical wizard page. A wizard will set the same parameters as available in the dialogs, only now they are presented in a sequential series of screens, and only the most important parameters are available. You can find the wizards via the Start Screen or via the File menu.

[2] Dialog-based interface.
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**. These set 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 (name.dpj), 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 illustration 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 Free edition only offers basic CAM functionality, Entry and Expert contain subsets of the available parameters, Multi-Axis includes all functionality. For the rest all editions are equal.

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.

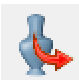
The functions of DeskProto can be reached using the pull-down menus or using the buttons on the toolbar. The most important menus are described below:

* The View Menu offers the opportunity to change what data should be displayed on your screen and how that should be done. Also try changing your view by rotating the six colored thumbwheels on the screen, and by using your mouse inside the view window. In fact most of the functions in the View menu can be activated most easily by using the button bar.

* In the Parameters Menu you can edit all vector/geometry/bitmap parameters and all milling parameters. For simple parts 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.

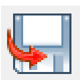
The most important buttons for the DeskProto process are:

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

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

Optionally you can show a **Simulation** of the result,
to check your toolpaths for correctness.

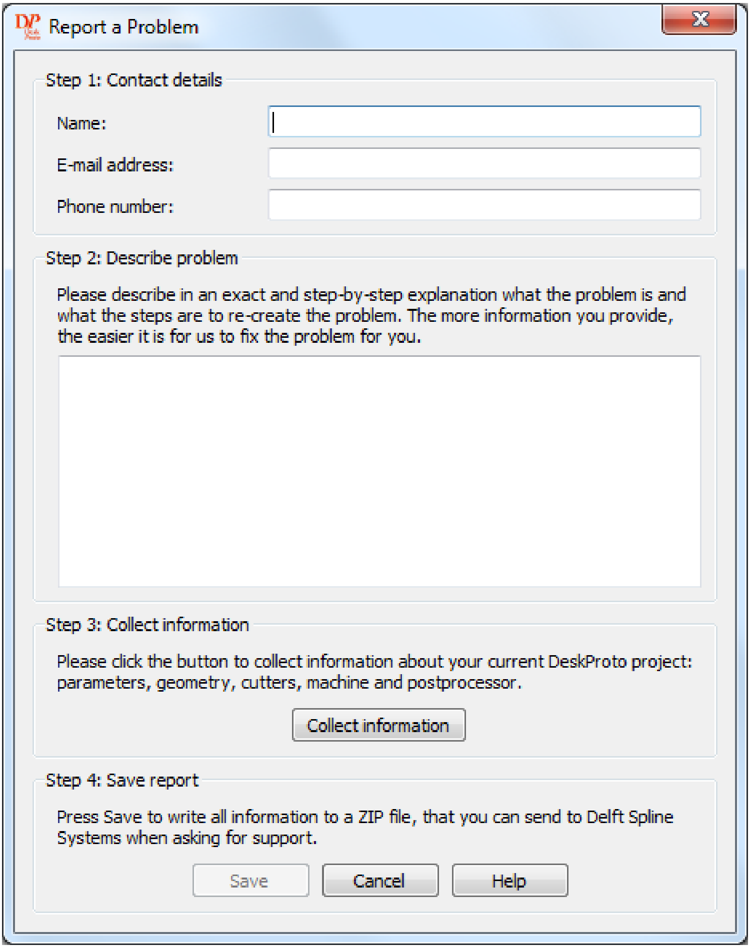


Finally you **Write the NC file**
and send it to your CNC milling machine.

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

3.7.3 Create Problem Report

When asking for support, DeskProto users often send us the DeskProto project file (DPJ file), intended to make us see what they are doing. Unfortunately this is not sufficient: this DPJ file only contains the parameters that they have set. And in order to reproduce the problem also the CAD file(s) are needed. Plus the driver files for cutters, machines and/or postprocessors. The Create Problem Report option automatically bundles all these files in one ZIP file, that you then can send to us.



Creating a problem report is done in four steps:

Step 1 asks you to enter your **Contact details**: information about how we can reach you (we ask only the bare minimum).

In Step 2 you are asked to **describe the problem**, in plain text. The more clear you describe the problem, the more chance that we can indeed reproduce it. And that is absolutely needed in order to start looking for a fix.
Note that you cannot actually use DeskProto when this Problem Report dialog is opened, so you cannot walk through all steps and type them during

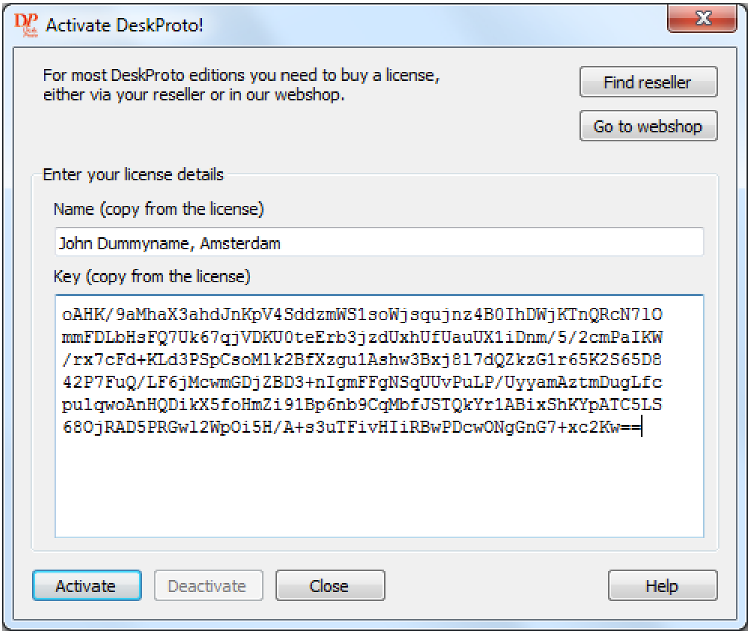
the process. If that is what you want you need to open Notepad, and after finishing copy the text from Notepad to this dialog.

In Step 3 DeskProto will **collect all information** needed to reproduce the problem: all files as just mentioned, and in addition some basic information about the Windows version and the graphics card that you are using.
Step 3 can only be started after you have entered the information asked for in steps 1 and 2.
As one of the files to be collected is the DPJ file, DeskProto will ask you to fist save this file (in case needed).

Step for finally **saves the report** file: a ZIP file that you can email to us.
Default name for this file is DpReport_Name.ZIP in which Name is the Name that you just entered.
Default location is your current directory, so the folder where you just loaded or saved your geometry and/or project.
You are however free to change file name and file location.

This same ZIP file can of course be used for other purposes as well, like sending a complete project to some other DeskProto user, or for archiving purposes.

3.7.4 Activate license



The **Free** Edition of DeskProto may be used without buying a license. Users who need more functionality can buy a license to activate (unlock) one of the higher Editions: **Entry**, **Expert** or **Multi-Axis**. The Free edition allows you to evaluate (trial) these higher Editions, though when trialling watermark (the "Trial cross") will be visible in all machined parts.

In all cases you need to first download the DeskProto Setup file from our website www.deskproto.com and instal DeskProto on your computer. This Setup file is the same for all Editions of DeskProto.

A **free DeskProto license** is given to any user: you are welcome to use the Free Edition of DeskProto free of charge.

A **paid DeskProto license** can be bought either via a reseller or via our webshop. After buying you will receive the license (a PDF file) containing two strings: a Name and a Key.

The **Name** contains the name of the buyer (either a person or a company) and his/her/its location (city, village): information that will be clearly shown at each program start.
The **Key** is a code of 340 characters, containing information all license information. Each Key is valid **only** for the Name on that same license.

The Activate dialog is meant to activate a DeskProto license and unlock the extra functionality for the Edition that you have bought. Both the complete **Name**-string and the 340 character **Key** have to be entered ***exactly as given***, including case (upper or lower), spaces, commas, points, etc. Any small difference will make DeskProto refuse to activate. So use Copy/Paste to enter this information in this dialog.
After filling both fields you can use the **Activate** button to make DeskProto activate the license.

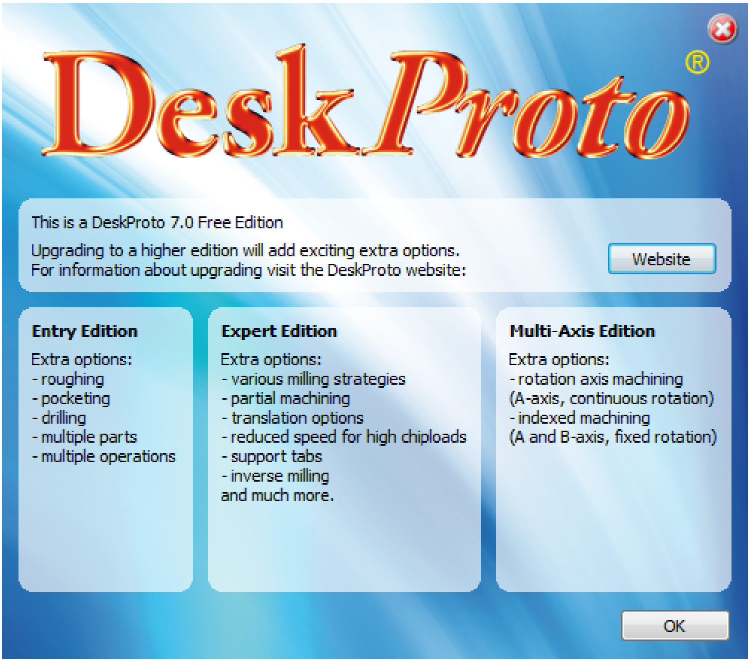
Make sure to carefully **save and backup** the license: you will again need Name and Key when (for instance) you buy a new computer.

A registered version of DeskProto will clearly show the Name of it's owner: in case that is not you then you are running an illegal copy!

After Activating it is also possible to **Deactivate**, allowing you to again convert the program from paid license to free license.
You my Activate and Deactivate as many times as you like: information about the previous licenses will simply be deleted.
Activating a new license is also possible without Deactivation first.

Both for Activating and for Deactivating **administrative privileges** are required.

3.7.5 Upgrade information



Displays a dialog screen that lists the advantages of the higher DeskProto Editions
(only shown in the Free Edition, the Entry Edition and the Expert Edition).

DeskProto is available in four different Editions:

1. **Free**
2. **Entry**
3. **Expert**
4. **Multi-Axis**

The difference between the first three editions is the number of parameters that you can set (both Part parameters and Operation parameters), and (for the Free edition) the number of Parts and Operations.

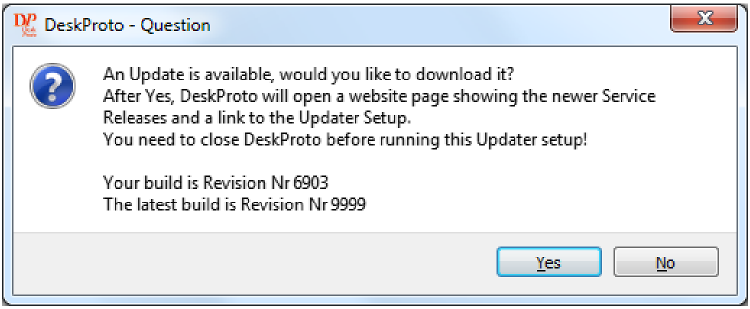
The difference between the Expert Edition and the Multi-Axis edition is that (of course) more axes are available in the latter: the A-axis and B-axes rotation axes.

Of course for registered users of DeskProto such upgrade is available at a special upgrade price.

3.7.6 Check for Update

This command will make your DeskProto contact the website www.deskproto.com (internet connection required) to see if a newer Service Release (bugfix build) is available for download.

Once every few months a newer build of the program is released, containing a number of bugfixes.
You can find the Revision number of your DeskProto in the About box.
'And you can find information about the Revisions that have been released on
www.deskproto.com/support/buildhistory.htm



The command Check for Updates will automatically compare the Revision number of your DeskProto with the Revision number of the latest release, will tell you when a newer release is available for download, and will ask if you ant to download it. When you answer Yes DeskProto will open the page on the DeskProto website where you can download the **Updater** Setup.

Running the Updater Setup will recognize your DeskProto license and keep that active.
Only the program will be updated: drivers, scripts, language files etc will remain as they were.

When no internet connection is available DeskProto will tell you that it "Could not perform the Update check".

3.7.7 About DeskProto

The DeskProto About Box will display information about the DeskProto version that you are running:

Version number, like 7.0

Edition - so either Free, Entry, Expert or Multi-Axis

Build date - the format of the build date is “yyyy-mm-dd”, and it show that date on which your file DeskProto.exe has been built.
and **Revision number** - each build of the file DeskProto.exe receives a new revision number.

Serial number - Each new license gets a new number, so the Serial Nr of your license is unique (empty for the Free edition).

Licensed to - the Name on which this license has been registered (empty for the Free edition)

Copyright notice - all rights reserved. You do not own the software: you own a license to use the software

Contact information - DeskProto is made by Delft Spline Systems in the Netherlands

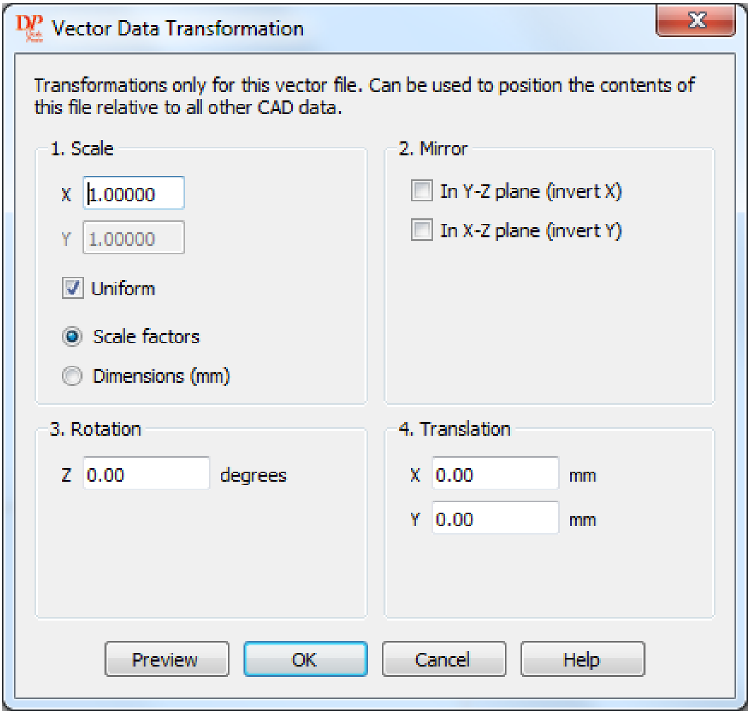
Short program description.

At the bottom of this dialog, next to the OK button you can find the button **Show Copyrights**.
Pressing that button will show the license texts for all external libraries that are used in DeskProto.

IV Dialogs

4.1 File

4.1.1 Vector Data Transformation



DeskProto V7 allows loading more than one Vector data file and more than one Geometry file. For each file DeskProto simply reads the coordinates in the file (as defined by the CAD systems that wrote each file) and draws the CAD data using these coordinates.

This may result in unwanted collisions and/or overlaps. Such unwanted situation can be corrected as you can give each file it's own set of transformations. For instance a translation to position two identical curve

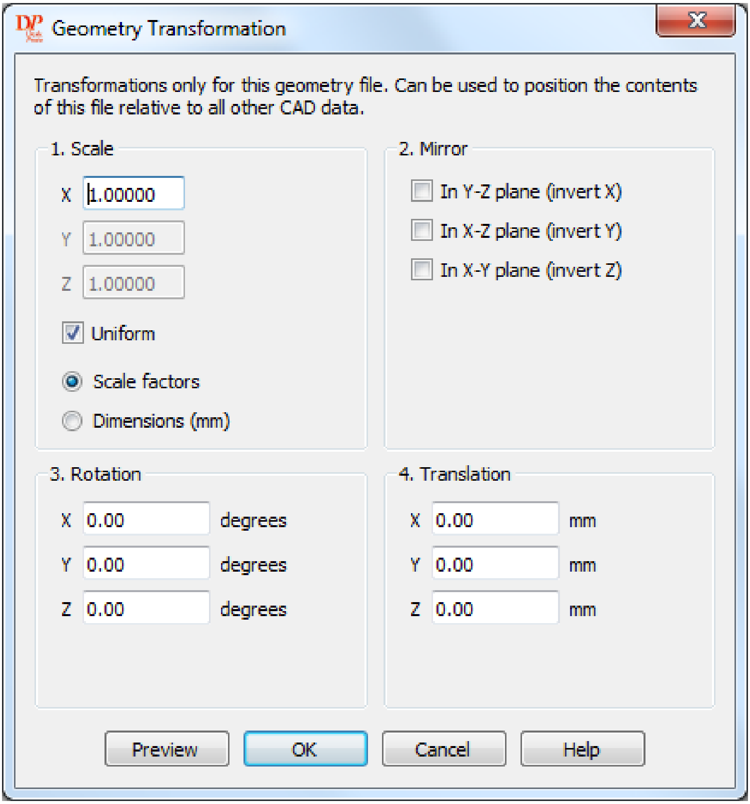
sets next to one another. In the **Vector Transformation** dialog these Transformations can be set for a Vector Data file.
For Vector files that also contain Z-values a third entry (Z) is present for Scaling, Mirroring and Translating.

This dialog can be opened on the Vector page of the Project parameters, using button Transform....
It will also pop up when loading an extra Vector file, and enables you to position the new vector data relative to CAD data that is already present. It offers basically the same transformations as also present in the Part Parameters (see there for more help), however these will be applied to the new vector file only.

A very handy tool in this dialog is the **Preview** button, as it will draw the new vector curves according to the transformations that you entered in the dialog. If this new position is not OK, you can enter new transformation values and again press Preview for a new drawing. When all is OK you can acknowledge with OK. Cancel will not cancel adding the new vector curves, only the (changed) transformation settings for this set of curves.

For visual feedback you need to realize that in the drawing that you see the Part Transformations of the visible part have been applied as well. This means that when a rotation is applied in the Part Parameters, it may be difficult to find transformation to produce the desired result.

4.1.2 Geometry Transformation



DeskProto V7 allows loading more than one Vector data file and more than one Geometry file. For each file DeskProto simply reads the coordinates in the file (as defined by the CAD systems that wrote each file) and draws the CAD data using these coordinates.

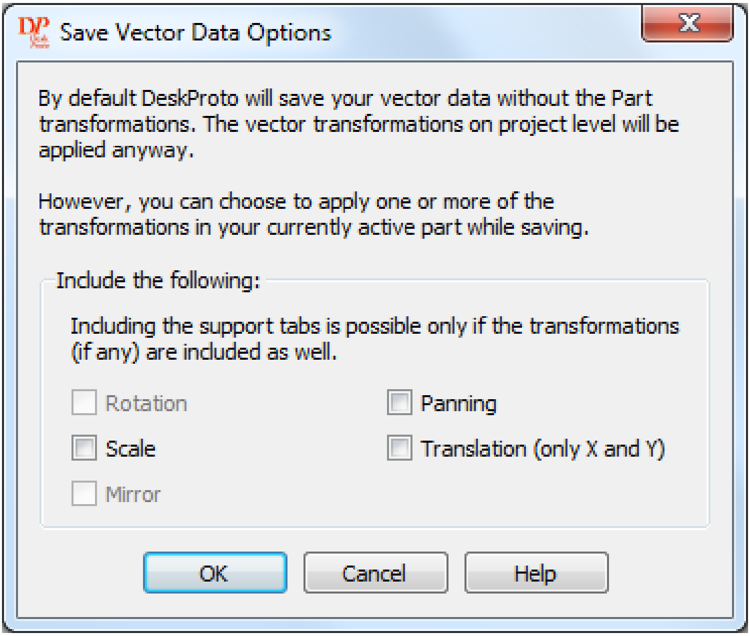
This may result in unwanted collisions and/or overlaps. Such unwanted situation can be corrected as you can give each file it's own set of transformations. For instance a translation to position two identical geometries next to one another. In the **Geometry Transformation** dialog these Transformations can be set for a Geometry file.

This dialog can be opened on the Geometry page of the Project parameters, using button Transform....
It will also pop up when loading an extra Geometry file, and enables you to position the new geometry relative to CAD data that is already present. It offers basically the same transformations as also present in the Part Parameters (see there for more help), however these will be applied to the new geometry file only.

A very handy tool in this dialog is the **Preview** button, as it will draw the new geometry according to the transformations that you entered in the dialog. If this new position is not OK, you can enter new transformation values and again press Preview for a new drawing. When all is OK you can acknowledge with OK. Cancel will not cancel adding the new geometry, only the (changed) transformation settings for this geometry.

For visual feedback you need to realize that in the drawing that you see the Part Transformations of the visible part have been applied as well. This means that when a rotation is applied in the Part Parameters, it may be difficult to find transformation to produce the desired result.

4.1.3 Save Vector Data Options

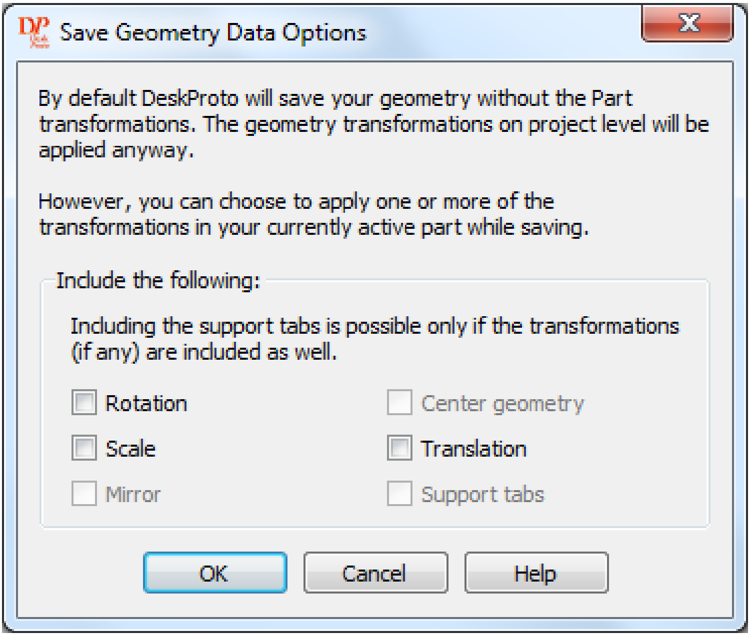


The **Save Vector Options** dialog enables you to apply Transformations that you have set in the Part Parameters. It will pop up after pressing command Save Vector Data As... in the File menu.

It offers the five Vector transformations that can be set in the Part Parameters (see there for more help). Note that only those transformations can be checked that indeed have been applied. When you check a transformation, DeskProto will save the Vector data including that transformation. So the new file will then contain rotated curves, scaled curves, etc. This is ideal for instance for saving a scaled or rotated version of your CAD file.

In case multiple vector data files have been loaded the total of all vector curves. will be exported.
The Vector Data Transformations that have been set for each file on Project level in all cases will be applied when exporting a Vector data file.

4.1.4 Save Geometry Data Options

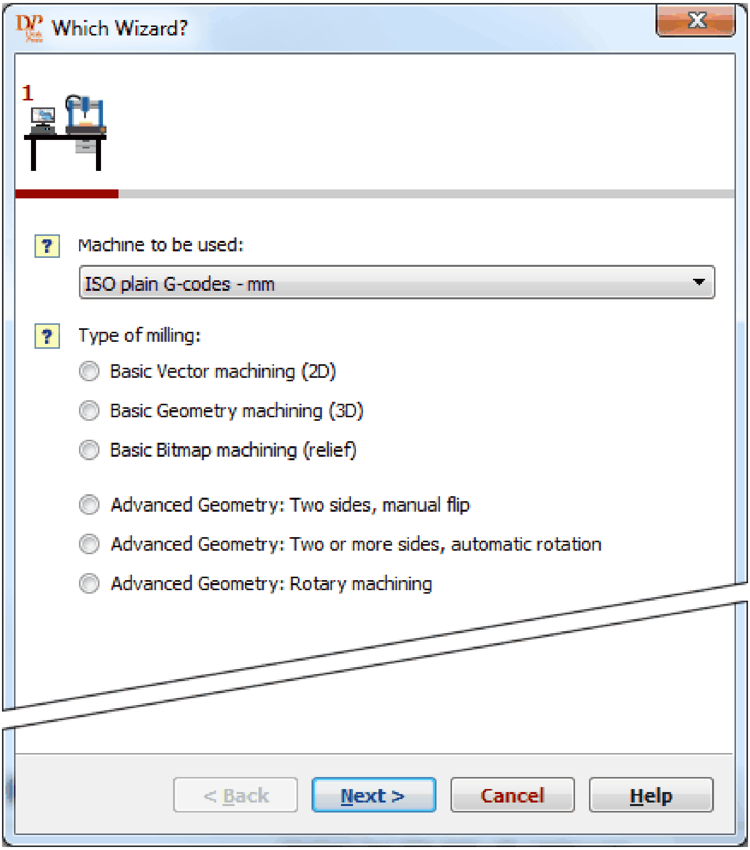


The **Save Geometry Options** dialog enables you to apply Transformations that you have set in the Part Parameters. It will pop up after pressing command Save Geometry Data As... in the File menu.

It offers the six transformations that can be set in the Part Parameters (see there for more help). Note that only those transformations can be checked that indeed have been applied. When you check a transformation, DeskProto will save the 3D geometry including that transformation. So new file will then contain rotated geometry, scaled geometry, with support tabs, etc. This is ideal for instance for saving a scaled or rotated version of your geometry file. This feature also can be used to create more than 4 support tabs (save with 4 tabs, load again and again add tabs).

In case multiple geometry files have been loaded the total of all geometries will be exported.
The Geometry Transformations that have been set for each file on Project level in all cases will be applied when exporting a Geometry file.

4.2 Wizards



The Wizard-based user interface is an important feature of DeskProto. It makes it possible for users without CAM know-how to easily create the toolpaths that they need for their projects. Each wizard is a series of dialog screens (forms) that need to be filled in: the wizard in a way *'takes your hand'* and guides you through the process of creating toolpaths.

When you create a project using a wizard and you save the project without making any changes after finishing the wizard, on re-opening DeskProto will show the project using the same wizard. So it is possible to use DeskProto seeing only the wizard interface. This is new in DeskProto V7, and will be a great help for inexperienced users.

Navigating the Wizard is done using the Wizard buttons, on the bottom of the screen:

Back and


Next let you navigate the wizard pages,

Cancel and


Help are standard buttons. Every wizard page has it's own Help information page.

Finish will replace Next on the final page of the Wizard,

Apply will refresh your screen and apply the currently selected settings,

Calculate will calculate the toolpath with the currently selected settings.

Navigation icons: The DeskProto wizards start with the page shown above, called "Which Wizard?". On the top of the page you can see one icon, after selecting one of the wizards a series of icons will be shown: one for each wizard page. At this point these extra icons are all gray, when progressing they will be colored: then they can be used as tab-pages to navigate (click on a colored icon to jump to that page of the wizard).

Help information: Important are the  yellow question marks in front of each question. When you position the cursor over such mark a Tooltip will pop up, giving extra information about that question. It will also tell you where to find that setting in the Dialog based interface.

The first question, "*Machine to be used*", in most cases needs not be changed as your default machine will already have been selected here.

The second question, "*Type of milling*", presents the six wizards that you can choose from: you need to select one before continuing.

Not all wizards will be available for all users (some may have been grayed out): some wizards are not available in all DeskProto **Editions**, and some wizards are only available in case you selected a machine with a rotation axis. Also: in the Free edition and the Entry edition the available wizards do not include all described options.

Basic Vector Machining
This wizard creates 2D toolpaths for just one Vector file (DXF, EPS, AI). It is meant for novice DeskProto users, and explains this procedure step-by-step. You can choose to use either Profiling or Pocketing toolpaths.

Basic Geometry Machining
This wizard creates 3D toolpaths for just one Geometry file (STL, DXF). It is meant for novice DeskProto users, and explains step-by-step the procedure to create an NC toolpath file (NC program) based on your geometry. The model will be machined from one side, using three operations: Roughing, Finishing and a smoothing contour.

Basic Bitmap Machining
This wizard creates 3D toolpaths for a relief based on a Bitmap file (BMP, GIF, JPG, PNG, TIFF file). It is meant for novice DeskProto users, and explains this procedure step-by-step.

Advanced Geometry (the three advanced wizards are all for geometry machining):

Two sides, manual flip, also called **Two-sided Wizard** (not available in the Free Edition and the Entry edition)
This advanced wizard is a unique feature of DeskProto, and makes it very easy for you to create a compete 3D part by machining it from two sides, on any three axis milling machine. DeskProto assists you by taking care of the repositioning needed to machine the second half: no need to change the workpiece zero point (starting position of the cutter).

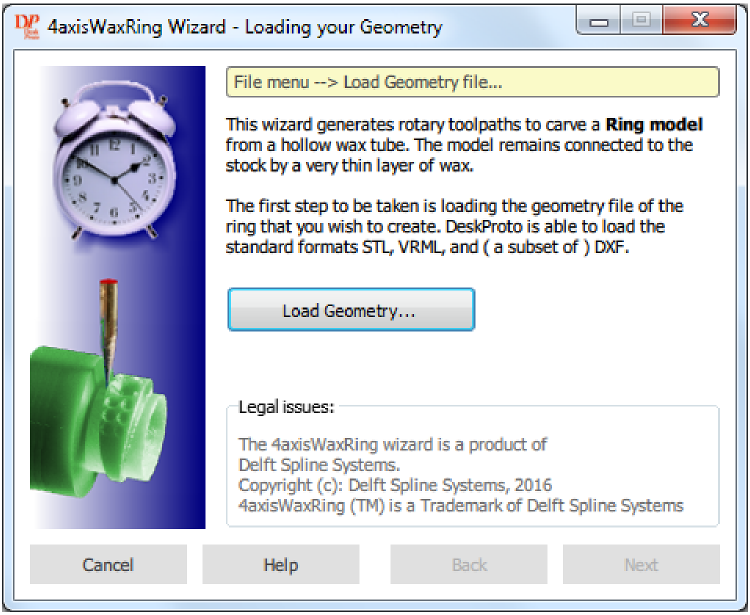
Two or more sides, automatic rotation, also called **N-Sided Wizard** (available only in the Multi-Axis edition)
N-sided milling is meant for machines with a rotation axis (A-axis), and allows **indexed machining**: this wizard generates toolpaths to machine the part from several (N) sides, with a rotation in-between. The number of sides can be freely chosen: for two sides the result is the same as for the previous wizard, though now with automatic rotation.

Rotary Machining (available only in the Multi-Axis edition)
If your machine is equipped with a rotation-axis (A-axis) you can use this wizard to create toolpaths for a model that is machined from all sides: rotation axis machining. In contrast to the previous wizard, now the material rotates during machining.
A rotation axis is an extra piece of equipment that lets your model rotate during machining (just like the meat rotating on a spit above a barbecue).

Note that all functionality offered by the wizards is also available in the dialog-based user interface: the wizards are only meant to make things easier for you, they do not add new options. After finishing any wizard you can still use the dialogs to fine-tune the settings that the wizard made.

You can find the Wizard in the File menu (File >> Start wizard) or in the Start Screen.
A keyboard shortcut is available as well: Ctrl + W.

4.2.1 Custom Wizard Page



Custom wizards can be added by any user, as explained on the Custom Wizards page.
Because of that in the Help file no Help pages are available for these wizards: on all custom wizard pages the Help button will show this same help page.

For all pages the four buttons at the bottom of the page will be the same, the rest of the dialog has been filled by whoever made this custom wizard. So in case of any question read the wizard's documentation, or contact the supplier of the wizards.

4.3 Parameters

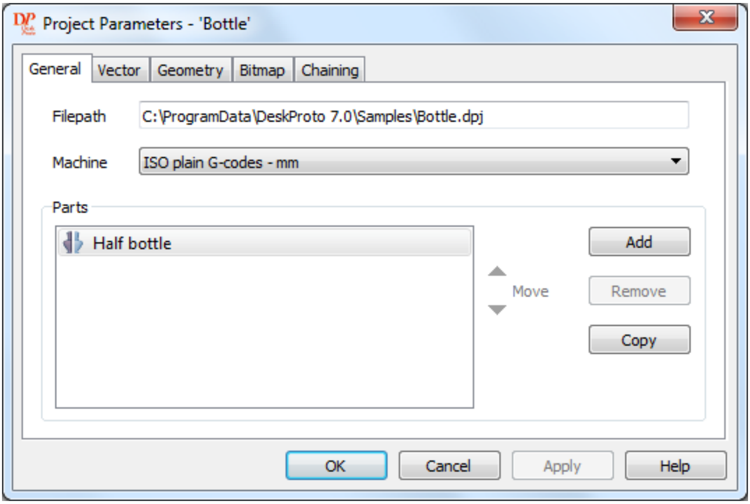
4.3.1 Project Parameters

The Project parameters dialog offers all settings that are available on project level. Most important are the CNC milling **machine** to be used and the CAD **data files**. Each of the five tab pages will be described in detail below.

Note that the Project Parameters dialog has an extra button: Apply. Using this button you can apply any new setting immediately, without having to first close the dialog. Any changes will be immediately reflected in the drawing on screen and in the Geometry information dialog.

This dialog can be reached via the Parameters menu, or by double-clicking the Project line in the Tree
The same dialog is used for the Default Project parameters.

General parameters



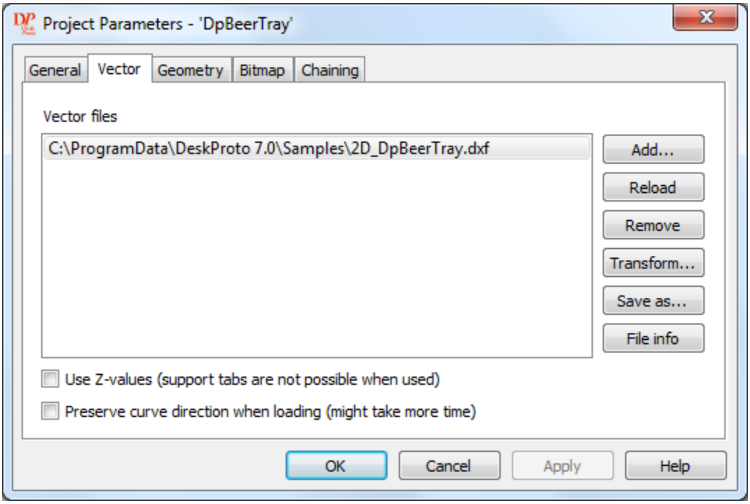
Filepath
This filed shows the path and name of the currently loaded Project file. This information cannot be changed in this dialog. To start a new project or to save the project using a different name, see the File menu.

Machine
Here you can select the machine you want to use. DeskProto will use the machine information to check if the part is not too large for the machine, if the speeds that you enter are possible for this machine, if the number of axes is available. The selected machine also defines the format of the NC Program file, because the machine information states which postprocessor (driver) to use. You can have a look at the machine-definition using the option Library of machines in the Options menu.

Normally you do not have to bother with this parameter as your machine will be the default machine (as set the first time that DeskProto was started, in the Initial Settings). If not, you can change the default machine using the option Default Project Parameters in the Options menu.
Note: when you change the selected machine DeskProto will automatically change values for Feedrate and Spindle speed for all operations in the project, as the new machine may not be capable of outputting the old values.

Parts
The number of Parts present in the project can be controlled here, using the buttons **Add**, **Remove** and **Copy**. Multiple parts can be used in a project to create different models, for instance a scale model and a full size, or a front part and a back part.
You can also influence the sequence of the parts, using the two **Move** buttons. The part that is selected (it's line made blue) will be moved in the direction of the arrow. This does not have any effect an the resulting toolpaths, it can be handy though in order to neatly arrange your parts in case you have many.

Vector parameters



Vector files

This tab page shows all Vector files (if any) being used in this project. When more than one file is present only one file is selected in this dialog (blue background color): the buttons on the right will influence that file (except the Add button of course).

You can change the set of files using the **Add** button and the **Remove** button on the right, to add and remove files.

When a vector file has been changed (for instance again exported from CAD) you can **Reload** it.

Very powerful is the **Transform** option: for each file you can define a set of transformations, for instance in order to position two vector drawings next to one another in 3D space.

Save As offers the option to save one vector file. The Save Vector data options dialog will op up, allowing you to select which transformations to use for the new file.

The **File Info** button finally shows information of the selected vector file: type, size, number of curves, dimensions.

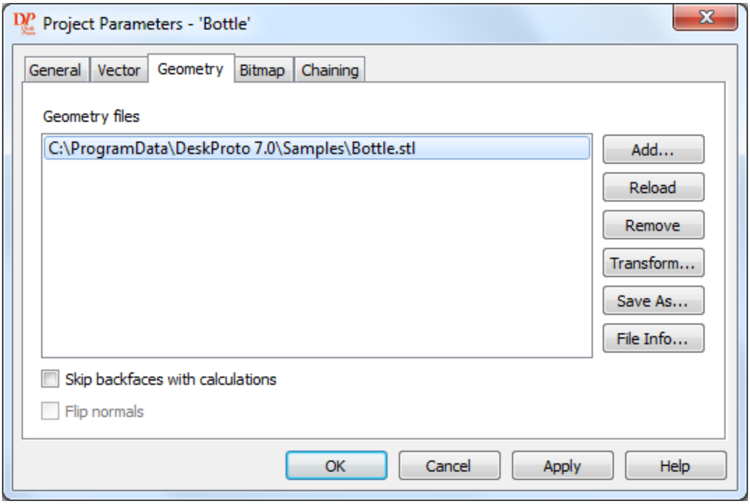
Use Z-values

Vector machining in most cases will be done using two-dimensional CAD data: a drawing that does not contain any Z-values (only X and Y, to be shown at Z=0.0). Nevertheless DeskProto will also accept vector files that contain Z-coordinates. When opening such file DeskProto will ask you whether or not you want to use these Z-coordinates. With this checkbox you can later change this setting.

When you use the Z-coordinates, the machining depth that you set in the Vector operation parameters will be taken relative to the Z-level of the curve and or point as defined in CAD.
When using Z-values, not all options in the Vector operation will be available. This option can also be checked for vector files that do not contain Z-values: DeskProto then will use Z=0 for all points on the vector curves. This will influence the result when both vector files and geometry files are loaded as the CAD-data zero level of the geometry file is used, which in most cases will not be the top of the block.

Preserve curve direction when loading
When loading a DXF file DeskProto checks if it can combine separate curves (polylines) to one curve: in case the endpoint of curve 1 is the same as the startpoint of curve 2 then for machining it is best to process them as one combined curve. This is especially important for recognizing closed curves.
When checking if two points are equal DeskProto even applies a small tolerance, in order to correctly process 'sloppy' drawings.
This option tells DeskProto what to do if two curves share either both endpoints or both startpoints: the curves then can be combined only if the curve-direction of one curve is reversed. As default setting DeskProto will indeed reverse curve direction in order to combine as many curves as possible; checking this option forbids DeskProto to do so. This may be handy in case you want to use the curve direction as defined in CAD for your toolpaths.

Geometry parameters



Geometry files

This tab page shows all Geometry files (if any) being used in this project. When more than one file is present only one file is selected in this dialog (blue background color): the buttons on the right will influence that file (except the Add button of course).

You can change the set of files using the **Add** button and the **Remove** button on the right, to add and remove files.

When a geometry file has been changed (for instance again exported from CAD) you can **Reload** it.

Very powerful is the **Transform** option: for each file you can define a set of transformations, for instance in order to position two geometries next to one another in 3D space.

Save As offers the option to save one geometry file. The Save Geometry data options dialog will op up, allowing you to select which transformations to use for the new file.

The **File Info** button finally shows information of the selected geometry file: type, size, number of facets, dimensions.

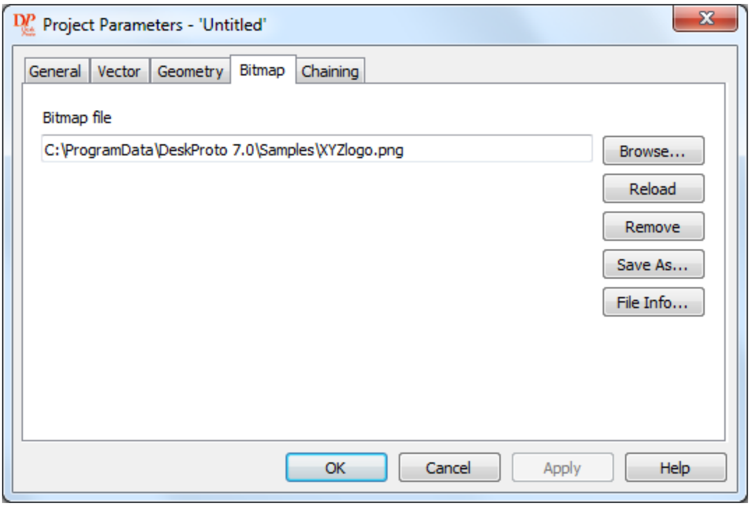
Skip Backfaces

For heavy calculations you may want to check the option Skip Backfaces in order to save calculation time. This option will skip all facets (triangles) of which the invisible inside (the backface) is on top as seen from the positive Z-direction. This means that half of the triangles can be skipped, which results in shorter calculation time. The resulting toolpath will be the same (for

a valid geometry). As on current computers the difference in calculation time is not very large the option is no longer checked by default.
For corrupt geometries you may need to disable this function.

Flip Normals
In DeskProto for each facet (triangle) a normal vector is stored, indicating which side of the facet is on the outside of the geometry (also see the previous paragraph, on Backfaces). This information is stored in the STL file. When using a corrupt STL file this information can be wrong. In case ALL normal-vectors point to the inside of the geometry you can use this option Flip normals to correct the normal information. This option is available only when Skip Backfaces has been checked, as otherwise the normal direction is not used anyway.

Bitmap parameters

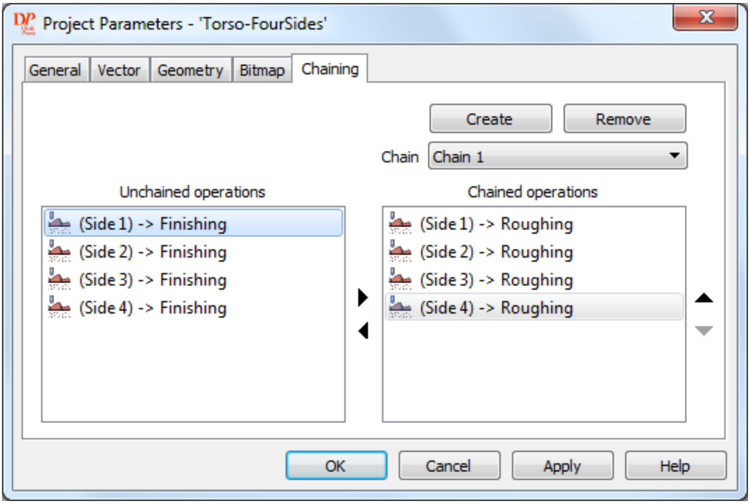


Bitmap file
This tab page shows the Bitmap file (if any) being used in this project (for bitmap data maximum one file can be loaded in a DeskProto project).
You can change this file using the **Browse** button, on the right, and remove it with the **Remove** button.
When a bitmap file has been changed (for instance again exported from CAD) you can **Reload** it.
Save As offers the option to save the bitmap file.

The **File Info** button finally shows information of the selected bitmap file: type, size, number of pixels, DPI, dimensions.

For bitmap files no extra options are available.

Chaining parameters



Chaining is an option meant to combine Operations in **various Parts** into one large NC Program file. Operations in the same Part will be combined into one NC Program file anyway, so chaining is not needed then. This option is used when more than one part is involved, for instance by the N-sided milling wizard, for indexed machining.
Note that chaining is of course only available in case more than one Operation is present in the current project.
Also note that Chaining is an option for advanced users.

The button **Create** adds an empty new chain to the list of chains.

The button **Remove** deletes the currently selected chain.

The combo box **Chain** shows the currently selected chain (if any), and makes it possible to select other chains. Chain names are automatically generated: Chain 1, Chain 2, etc.

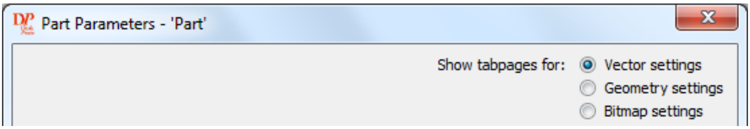
The field **Chained Operations** shows all operations in the currently selected chain. You can use the **up and down buttons** to change the sequence of the operations in the chain.

The field **Unchained Operations** shows all operations in the project that are not (yet) part of a chain. You can use the **left and right arrow buttons** to move unchained operations to a chain and to remove operations from the selected chain.

In the Operation Start/End commands you can add extra commands to be executed in-between the two chained operations: either after the first Operation or before the subsequent operation. For instance to rotate the rotation axis to the correct angle before starting that operation.

4.3.2 Part Parameters

The Part parameters dialog offers all settings that are available on part level. The parameters are applied to the CAD-data in the sequence as presented by the Tab pages, from left to right. The further to the right, the more advanced the parameters in the tab. Each of the tab pages will be described in detail below.



The number of tabs that is present is different for each type of CAD-data, as each data type comes with different settings:
Vector Settings
Geometry Settings
Bitmap Settings
When more than one type of CAD-data has been loaded you can therefore select the set of settings that you want to edit, when no CAD-data is present only the tab General will be visible. Some of the tabs pages are identical for all data types, some are present for just one of the three types. Every tab page description below mentions for which type of data it is present.

The number of tabs that is present also is different for each Edition.

For instance in the Free Edition and in the Entry Edition only the first two Tab pages are available, as these editions offers less parameters than the other editions.

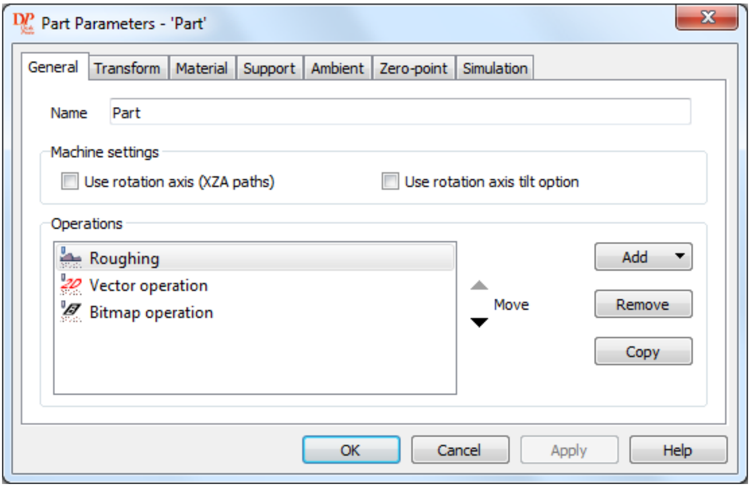
The Part Parameters dialog has an extra button: **Apply**. Using this button you can apply any new setting immediately, without having to first close the dialog. Any changes will be immediately reflected in the drawing on screen and in the Part information dialog.

This dialog can be reached via the Parameters menu.
Shortcuts:
You can double-click on a part-item in the project tree (one of the second level items).
Or right-click on a part-item and select Part Parameters in the context-menu.

This same dialog is used for the Default Part parameters, only with an extra button **Restore DeskProto defaults** to reset the original default parameters.

General parameters

(Vector, Geometry and Bitmap Settings)



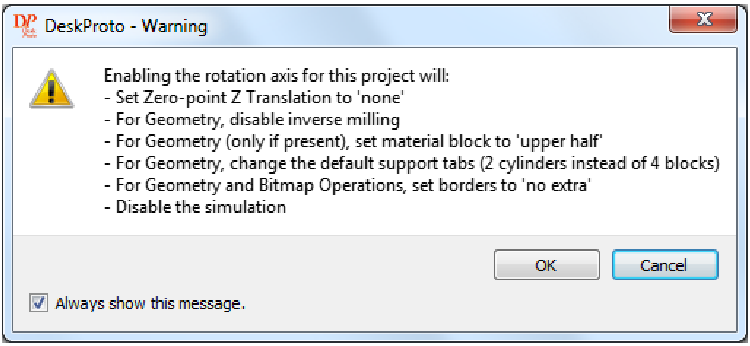
Name
The name of the part can be changed here. Use a meaningful name to easily remember the purpose of each specific part: the Project Tree then will be

easy to interpret in case of more than one part. This name is for your convenience only: it is not used in the resulting NC program file.

Machine settings
These are visible and/or enabled only in the Multi-Axis edition of DeskProto, and only if the machine with a Rotation axis and/or with a Tilt option has been selected in the project parameters.

Use rotation-axis
A rotation axis is a device on the machine that rotates the part during machining. Like a piece of meat on a spit above the barbecue. Note the difference between an A-axis and a lathe: on a lathe the spinning of the part causes the cutting, on a rotation axis the spinning of the tool.

By default DeskProto ignores the rotation axis and generates standard XYZ toolpaths.
If "Use rotation axis" is checked, DeskProto will generate XZA toolpaths instead, with A-value in degrees. In the View Window the difference will be immediately visible, as the material block no longer is a rectangular block, but a cylinder instead.
The part will rotate when machining: continuous rotation. This in contrast to the indexed machining that some CAM programs offer (which DeskProto also offers, in the N-sided wizard).
For geometry the part will also rotate the CAD-data in DeskProto (toolpaths from all sides), vector data and bitmap data will be wrapped around a cylinder.



When you check “Use Rotation axis”,DeskProto will automatically (when applicable):

- set the Zero-point Z Translation to "None" for the Z-axis (resulting in a Workpiece zero point exactly on the rotation axis)
- disable Inverse milling.
- set the Material block to “Use upper half of geometry”, in order not to let the tool sink below the rotation axis.
- change the default setting for the Support Tabs.
- set the Borders to "No extra" for all operations in that part (as the part must not be cut loose of the rotation axis)
- disable the Simulation, as DeskProto cannot simulate rotation axis toolpaths.

You are free to change these choices later in case needed.
The message box shown above will remind you of these changes, you can switch it off when no longer needed (it can be restored in the Preferences, tab Advanced, subset Settings).

By default DeskProto calculates for a rotation axis parallel to the X-axis, which is officially known as an A-axis. A rotation axis parallel to Y is supported as well: you can configure that in the Advanced machine settings.

Use rotation axis tilt option
This option concerns a special type of fifth axis: a mechanism that tilts the complete rotation axis unit. Such fifth axis is present on some machines offered by Roland (JWX-10, MDX-40), and is meant to machine the inside of a ring. In fact this is a B-axis. DeskProto does not control the movement of this axis: it has to be manually set. For more information and a picture see the Tilt option tab of the Operation Parameters. This tab page is present only if this tilt option is checked.

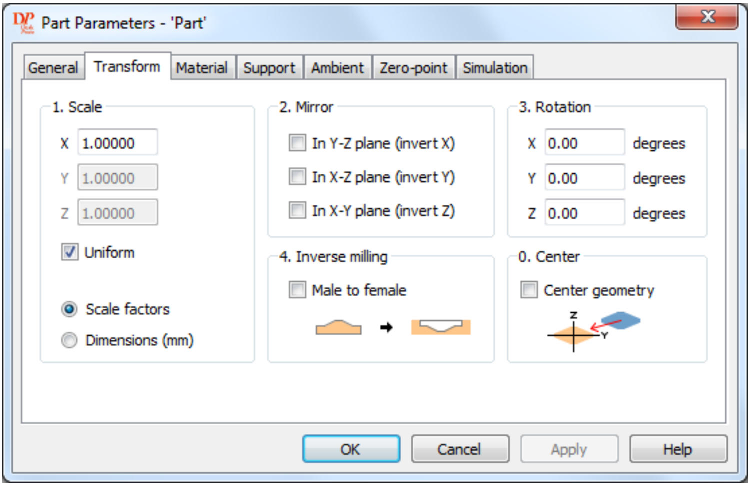
This option is only available in case a machine has been selected that supports this "Rotation axis tilt option" (in case the machine-definition in the Library of machines does so). This option also is available only in the DeskProto Multi-Axis edition. If this box is not checked DeskProto will ignore the tilt option and generates standard toolpaths.

Operations
The number of operations within the part can be changed by adding, removing or copying operations, using the buttons **Add**, **Copy** and **Remove**. As three different types of Operation can be added: Vector, Geometry and Bitmap, as the Add button comes in three flavors.
When more than one operation is needed to create a part (for instance first roughing with a large cutter and then finishing with a smaller one) the

sequence of the operations is important. Using the **Move** arrow buttons you can change the sequence of the operations.

Transform parameters

(Vector and Geometry Settings)



The availability and the contents of options 4 and 5 on this tab page will differ per edition of DeskProto (Free, Entry, Expert, Multi-Axis).
The options will also be different for Vector data and for Geometry data
The order in which the transformations are applied will affect the result, which is why they are numbered in the dialog.

1. Scale

Scaling is the first transformation that is applied to the original geometry / vector data. It is possible to scale differently for X, Y and Z, by un-checking the **Uniform** option. The imported CAD file does not contain information about the units used, DeskProto assumes the units used in the CAD file to be the same as set in the Preferences dialog. If this is not true it can be corrected using the scale factor:
Enter a scale factor of 25.4 to use a CAD file created in inches in a DeskProto that is configured to use metric units.
Enter a scale factor of 0.03937 to use a CAD file created in mm in a DeskProto that is configured to use inches.
You can scale either by using **scale factors** or by entering the desired new **dimensions**.

For Vector-data without Z-coordinates only X and Y can be scaled.

2. Mirror

The mirror option is the second transformation applied. It is only useful to mirror in one direction, as mirroring in two directions is identical to one rotation. Mirroring in three directions can be achieved by mirroring in one direction plus a rotation. The mirror option can be useful when you have a geometry that is one half of a prototype. By mirroring the geometry for the second part you can produce two parts that will exactly fit together.
For Vector-data without Z-coordinates only X and Y can be inverted.

3. Rotation

The rotate option is the third transformation applied. Note the difference between this rotation (which changes the geometry) and the view rotation (which only changes the viewpoint / the camera position). Both rotations use identical values, so you can use the Viewpoint to find the rotation you need, and then use these X, Y, Z rotation values to enter here.
When a custom Material block and/or Area is present (rectangular) only for rotations over (multiples of) 90 degrees the block or area can remain correct. For other angles the boundaries would be changed. For Freeform areas only rotations over (multiples of) 180 degrees are possible, other rotations are not permitted.
For Vector-data without Z-coordinates only rotations round Z are possible.

4. Inverse milling (Geometry settings)

The inverse milling option useful for producing a mold: a cavity in a solid block of material that exactly fits your geometry. In many cases it will be easier to create an inverse geometry using the original CAD system. However, this option comes in useful in case you only have the STL file, not the original CAD data.

Inverse milling is not the same as mirroring the Z-axis. A mold that is created by mirroring would produce mirror images of the original geometry. Instead DeskProto uses a 180 degree rotation to create the inverse. As this inverting is applied during toolpath calculations, the STL geometry on the DeskProto screen is not inverted. To get an idea of what you are machining you can display the rendered Z-grid or a simulation instead.

Note 1: when selecting this option, in most cases it is necessary also to change the Ambient and set it to 'Top level'.
Note 2: when this option is checked, setting the Z-values for the Material block and the Area to be machined still is done in un-inverted coordinates.

Inverse milling is not available in the DeskProto Free edition and Entry edition.

4. Panning (Vector settings)

Panning can be used to correctly position the Vector curves over a geometry and/or a bitmap. For instance when you want to engrave a logo on a 3D part. When you only have vector data loaded this option in most cases will not be used: for the NC file DeskProto will by default apply a Translation that will make the corner of the block the Workpiece Zero point (0.0, 0.0, 0.0)
The button **Align to...** leads to the Align dialog: a handy help to set a few useful locations.

The vector panning option will be disabled ("grayed out") when it does not make sense: in case only vector data is loaded AND the XY translation (tab Zero-point) is automatically calculated ("to positive XY" or "Make center"). The latter condition is always true in the Free edition and the Entry edition.

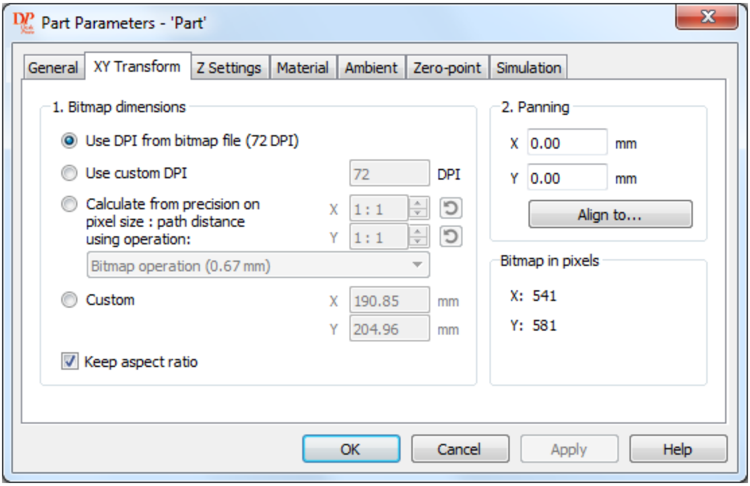
5. Center geometry (Geometry settings)

In DeskProto all Translations are applied at the end of the calculation pipeline, after calculating the actual toolpaths. For rotation axis machining this is not sufficient, as the position of the geometry during calculation influences the toolpaths that will result.
By default (when you do not check this option) during rotation axis machining DeskProto will rotate the geometry around the X-axis as defined in the CAD system.
When you check this option, DeskProto will rotate around a line parallel to the X-axis through the exact center of the part. You can immediately see the difference as this influences the size of the Material block: when checked a cylinder that exactly fits the geometry, when not checked a cylinder that is far too large if not.
When you need any other position of the rotation axis you have to go back to the CAD system and translate the geometry there.

In addition to this option you can still use the Translation options for a translation to be applied just before saving the toolpaths. For rotation axis machining you then can choose between a workpiece zero point on the rotation axis or on the top of the cylinder block (Z=0 level).

XY Transform parameters

(Bitmap Settings)



Bitmap dimensions

The dimensions on this tab page define the size of the Relief in XY, the size in Z is determined by the Z-settings for Black and for White.

- **Use DPI from bitmap file**. Most bitmap files contain a DPI value, which stands for Dots Per Inch. So for a 300 DPI image the size of one dot (pixel) will be set to 1/300 inch. As the number of pixels is known this value sets the exact size of the image, and thus for this choice the size of the relief. For files without a DPI value (for instance GIF files) DeskProto will use 96 as default.
- **Use custom DPI**. This options allows you to enter any DPI value. As explained above this value will set the exact size of the image, and thus also of the relief.
- **Calculate from precision...**. As explained in paragraph Bitmap Data DeskProto uses a Z-Grid to create the Relief. For options 1, 2 and 4 the ratio between Pixel size and Grid cell size (the path distance) is not necessarily a whole number. So it might be that some Grid cells 'contain' more pixels than others. This may lead to a **Moire pattern** like a small ridge every few mm. Which can be fixed by choosing this option and approximate the requested dimension as close as possible. One pixel will be converted to exactly ... , 1/4 , 1/3 , 1/2 , 1 , 2 , 3 , 4 , ... grid cells, as set by the ratio Pixel size / Path distance. As the path-distance is one of the Precision values in the Operation parameters, when more than one bitmap

operation is present you need to select the operation the you want to use. Important: when you later change the Precision, the size of the relief will change as well !

- **Custom**. Here you can enter any dimension as required.

The option **Keep aspect ratio** takes care that the ratio between X size and Y size is not changed: same ratio for both image and relief. So when you change the X size with this option checked, you will see that the Y size automatically changes too.

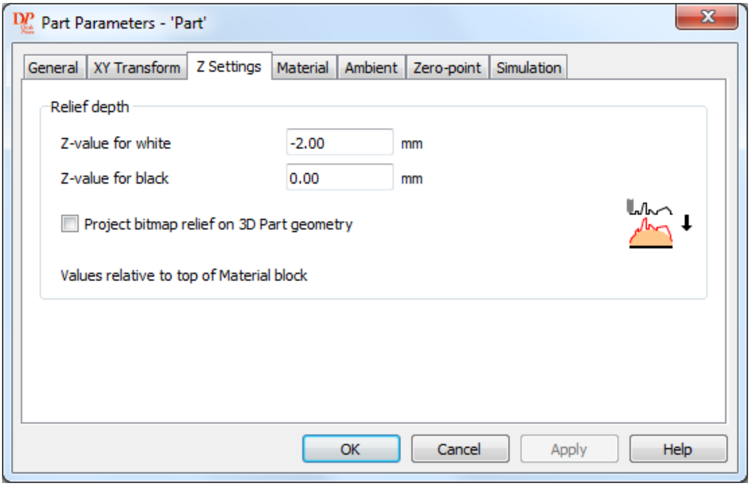
Panning

The position of the bitmap relief relative to the workpiece zero point can be changed here. By default the relief is located with it's lower left corner exactly at XY (0,0), so at the zero point (the Z depends on the settings of the second tab page). If this is not the correct location you can change it by entering Translation values for X and Y here. Panning can for instance be used to correctly position the bitmap relief over a geometry.
The button **Align to...** leads to the Align dialog: a handy help to set a few useful locations.

The bitmap panning option will be disabled ("grayed out") when it does not make sense: in case only bitmap data is loaded AND the XY translation (tab Zero-point) is automatically calculated ("to positive XY" or "Make center"). The latter condition is always true in the Free edition and the Entry edition.

Z Settings parameters

(Bitmap Settings)



In order to use a Bitmap image in DeskProto it needs to be transformed to a 3D Relief. As explained in paragraph Bitmap Geometry, this is done by created by calculating a Z-height for every pixel in the bitmap image, based on its gray value..

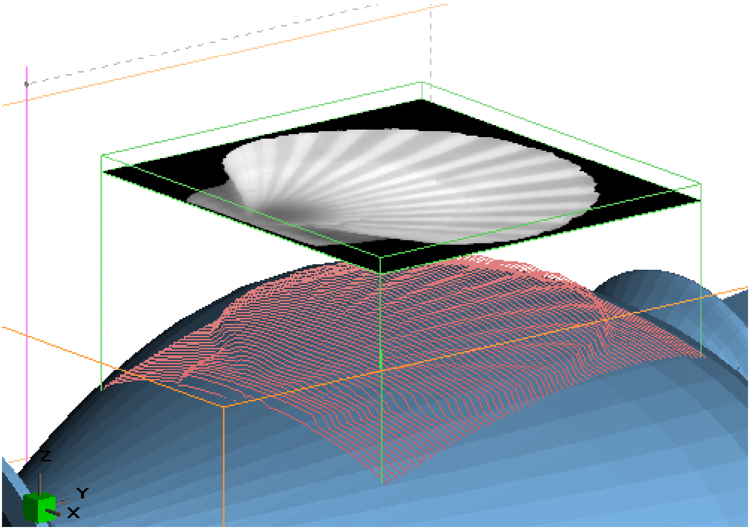
Z calculation

In these two edit boxes you can enter the Min and Max Z-values, to be used for black and white. DeskProto calculates this depth starting at the top of the block. So positive values do not make sense here (unless the option 'Project bitmap relief on 3D part geometry' has been checked).

In the image above the minimum Z for white pixels is -2.0 and the maximum Z for black is 0.0, which for gray values will result in the lighter the color the lower the relief (the other way round is also possible: black lowest and white highest). This is called **gray-value to Z-height conversion**.

Project bitmap relief on 3D part geometry (not in the Free edition and the Entry Edition)

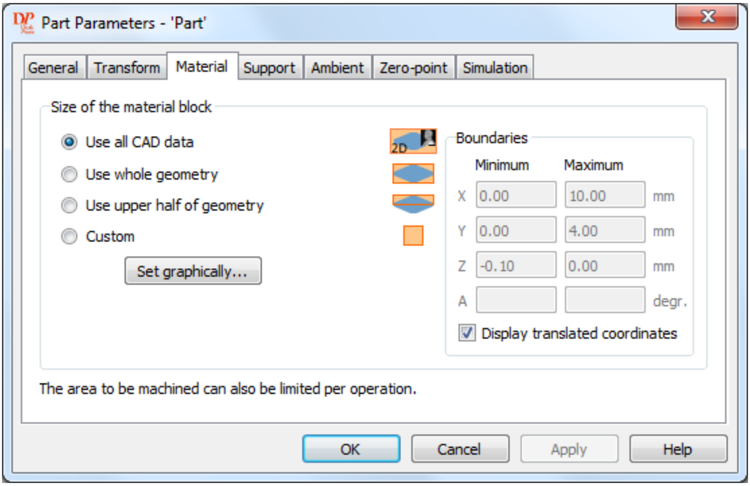
When this option is checked, the Z-value that is calculated for the color of that pixel is added to the Z-value of the geometry at that location. So the 3D relief is projected onto the underlying geometry, using a vertical projection. See the illustration below (where the relief depth for black has been set on 0.0 and for white on a positive value).



Here a shell relief is created from a bitmap image, and projected on the perfume bottle that you have seen more often as example part. This is of course a great design tool to create embossed products. The picture shows a positive relief: on top of the geometry.
Important for such positive relief is that of course material is needed to create this relief, so the previous operations used to machine the bottle may not machine the original bottle shape there. This can be achieved by making each operation a bitmap operation.
The relief to be projected can also be negative: when the Z-values range from 0.0 to a negative value, the relief will be subtracted from the part geometry.

Material parameters

(Vector, Geometry and Bitmap Settings)



The block of material to be used (the stock) is a rectangular volume, bounded by **Minimum** and **Maximum** values for each of the three axes. Only the CAD-data inside this block will be processed during the milling calculations, the rest will be discarded. As you can see in the icon pictures, the bounding box of the block will be drawn in **orange lines** (the green lines that you may see are for the Area to be machined).

The Min and Max A-values are available only in case you use a Rotation axis. The options that mention the Geometry are available only when a geometry file has been loaded.

Four options are available:

- **Use all CAD data** is the default: the block will be the bounding box that exactly fits all CAD data. For 2D vector files a block thickness of 10 mm will be used (minimum Z -10.0, for inch users -0.5"), unless the Machining depth as set in the Vector Operation parameters is more.
- **Use whole geometry** uses a bounding box that exactly fits the total geometry. In case you have only loaded one or more geometry files the result will be the same as for Use all CAD data.
- **Use upper half of geometry** is meant for symmetrical geometries that can be machined in two halves, for instance the right side and the left side of a power drill. Separately machine both halves and then glue them together to become one model.

- **Custom** is the option to define any other block size. In that case you can enter the block dimensions either by typing the appropriate minimum and maximum values, or using your mouse after pressing button Set Graphically. You can always reset the values by again selecting 'Use all CAD data. Note that the min and max values for the A-axis are in degrees instead of in mm or inches.

If you want you can enter the values for the block in **Translated coordinates**. When you have checked this checkbox the minimum and maximum values of the block (the boundaries) are displayed in the coordinates as used on the machine. These are the coordinates after Translation has been applied. A translation is also applied in the Free edition and the Entry edition (where no Translation tab is present): making the front/left/top corner of the block the zero point. Checking or unchecking this checkbox does not influence the toolpaths. It’s just a temporary conversion in this dialog to make setting the boundaries easier.

A custom material block size can be used when the block that you have is **larger** than the bounding box of the CAD data. For instance when your actual block is higher than the CAD geometry: the roughing layers will start counting from the top of the block, so the block thickness in DeskProto needs to match the real block. This can be done here by entering custom block dimensions.

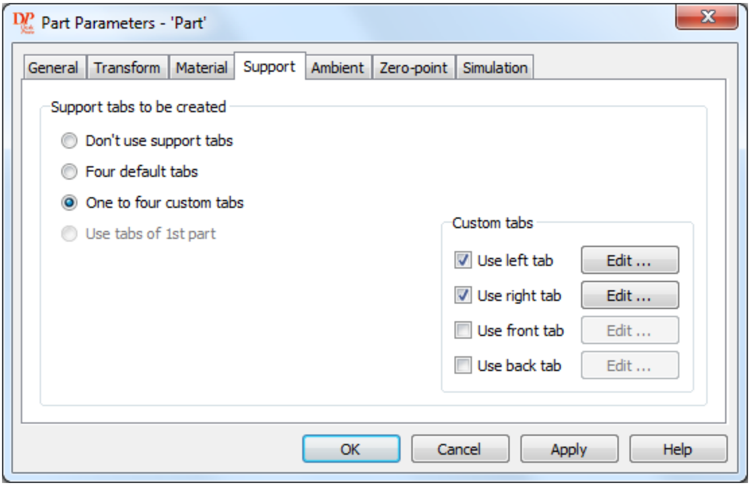
You can also use a custom block that is **smaller** than the CAD data. For instance when the part is too large for your machine. Say it is 500 mm long, while your machine can do max 300 mm (along X). Then in your DeskProto project define two parts. For the first part set a custom block with X-min 0.0 and X-max 250, and for the second part set a block with X ranging from 250 to 500. The two parts that you machine will exactly fit together to form the complete model. It will be needed to use the option Protect Vertical surfaces to make the splitting surfaces vertical.

The same trick can be used when your part is too high for your machine: build the model from separately machined slabs of material. For instance produce a model of 140 mm high, milling three separate slabs of standard tooling board blocks of 50 mm thick, using three parts with these material blocks: part1 Z 0-50 mm, part2 Z 50-100 and part3 Z 100-140 mm.

Note:
In case Inverse milling (male to female) is applied, then the block boundaries entered here are applied on the original geometry, before the male to female conversion.

Support parameters

(Geometry Settings)



Support tabs can be added to the geometry to keep the part connected to the rest of the block. These are small blocks of material (cylinder or rectangular), acting as connection bridges to hold the part on its place. Say your block is a slab that is much larger than the part: you then can clamp the slab on its corners and machine several parts, each on a different position, without the need to clamp each separate part.
The tabs are for instance needed for two-sided machining (see the Two Sided Milling Wizard), and for fixturing your model on a rotation axis device.

This tab page is about Geometry support tabs, made by adding small blocks to the geometry. Not to be confused with Vector tabs, made by changing the machining depth in a 2D toolpath.

Each support tab is created by loading an STL file from folder C:\Program Files\DeskProto 7.0\Supports\
The contents of this STL file will be scaled and rotated as needed, and will then be added to the geometry and be handled by DeskProto just like any other geometry detail.

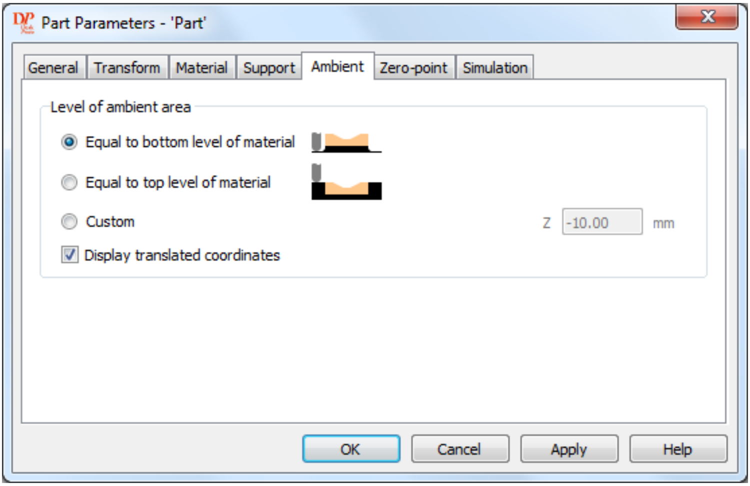
Four options are available:

- **Don't use** support tabs, which is the default and will be OK for most parts.
- Use **Default** support tabs: DeskProto will locate these tabs at the extreme points of the geometry (min and max X, min and max Y, applied after the rotation). For normal XYZ machining four rectangular tabs are generated, for rotation axis machining two cylinder shape tabs. Tab width and thickness depend on the size of the geometry, tab length is (at least) 1.5 times the cutter diameter (of which 10% sticks into the geometry), to make sure that the cutter can machine it.
- **Custom** tabs can be used when the default tabs do not satisfy, you can choose to create one to four custom tabs. For each of the four tabs you can then check whether or not to use them, and for each checked tab you can specify location and dimensions graphically, using the **Edit** button. This will open the Set support tab dialog, where you can also set the shape of the tab.
- **Use tabs of first part** is very convenient when several parts use the same support tabs. Define these support tabs only once (for the first part), and use that same definition in all other parts. In that case also any changes need to be made only once. This option is very handy for two-sided machining. For the first part obviously it will be disabled.

When using support tabs to connect the part to the remaining frame, you need to **switch off the borders** in each of the operations used for this part (Operation parameters, Tab page Borders). If not DeskProto will cut the outside surface of each tab as well, cutting it loose from the frame.

Ambient parameters

(Geometry and Bitmap Settings)



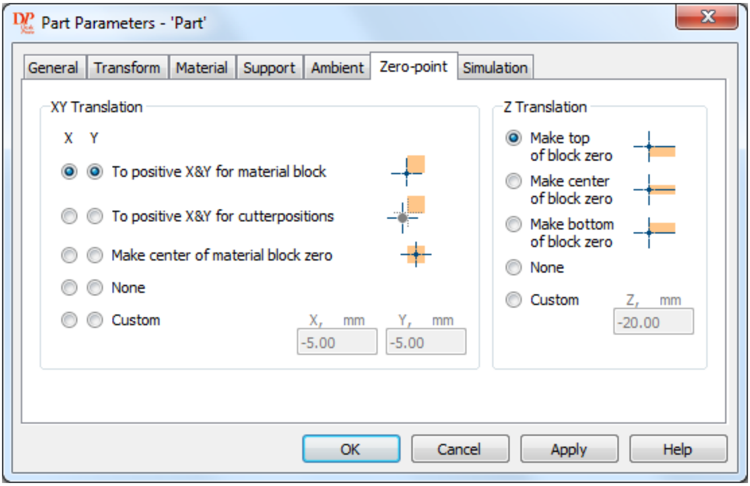
DeskProto toolpaths for Geometry and Bitmap by default will completely cover a rectangular area. Most geometries do not have a rectangular geometry though, so in many cases some area will be present without geometry. This is called the ambient area. Or in other words, the area where no geometry is present, seen from the top. On this tab page you can specify the level (height) at which this ambient area should be milled.

- The default choice is **Equal to bottom level of material**, so at the minimum Z-level of the block. This will do for most cases, as then all material around the part will be removed. **Note**: when using this option, for **ballnose cutters** the actual Z-level will be R (radius of the cutter) *lower than the bottom of the block*, to make vertical walls possible. So take care not to damage your machine's working table.
- **Equal to top level of material** will be useful when you want to make a mold by using inverse milling: then you do **not** want the material around the cavity to be removed.
- **Custom**, allows to specify any ambient level, by entering a Z-value edit box. **Hidden feature**: When you select this option, a ballnose cutter will *not* go R mm lower than the level in the Z edit box (as just explained for 'Equal to bottom'). So switching from "Equal to bottom" to "Custom" will not change the value in the Z edit box, however may nevertheless produce different toolpaths.

If you want you can enter the ambient level in **Translated coordinates**. When you have checked this checkbox the Z-level is displayed in the coordinates as used on the machine (after Translation has been applied). Checking or unchecking this checkbox does not influence the toolpaths. It's just a temporary conversion in this dialog to make setting the ambient level easier.

Zero-point parameters

(Vector, Geometry and Bitmap Settings)



Setting the **WorkPiece zero point** for your NC programs means applying a **Translation** to your CAD data: converting from CAD coordinates to machine coordinates. Although you can see translation as a form of transformation, it's not placed on the transformation tab page. In DeskProto the translation is applied after all actual calculations, just before saving the toolpaths to an NC-program. On screen the position of the WorkPiece zero point is shown by the **blue Orientator** cube (if not then you can switch it on in the Items Visible dialog).

Default translation is option 1 for X, Y and Z, making the front-left-top corner of the block the Workpiece zero point. This is customary for CNC milling and very handy: the zero position will now be with the tip of the tool touching the top of the material, at the front-left corner of the block. So all X and Y positions to be machined will be positive (starting at 0), and all Z positions will be negative (starting at 0 as well). Cutter positions may still have a negative X and/or Y value, as the cutter needs to move outside the block to machine the outer surfaces.

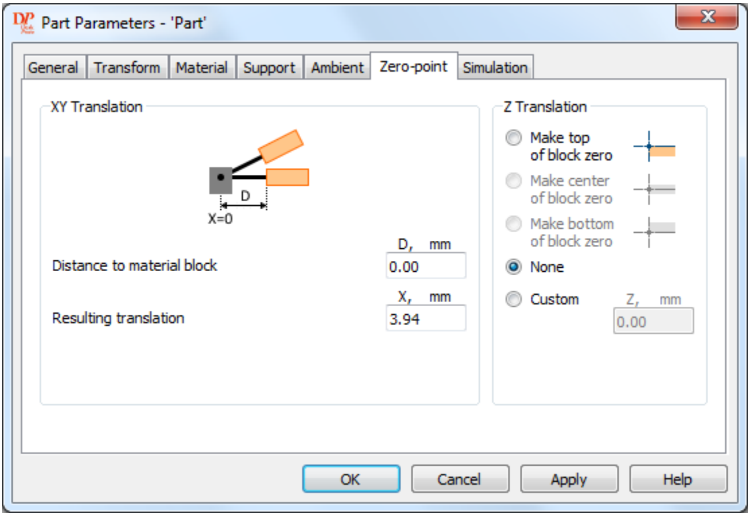
It is possible to use different settings for X, for Y and for Z. For each axis five predefined translation options are available. For each option DeskProto will calculate the actual translation values and show it in the edit box.

XY:
- Translate **To positive X&Y for material block** has just been explained: X=0.0 rep Y=0.0 will be on the edge of the block. This option is the most convenient one because it will be easy to set the zero-position on the machine.
- Translate **To positive X&Y for cutter positions** will make all cutter positions (or toolpath coordinates) have a positive X or Y coordinate. This option is useful in case the machine can handle only positive X & Y coordinates (for example the small Roland Modela MDX 15 and 20).
- **Make center of material block zero** is preferred by some users as their default option. It is a convenient option for two-sided machining using reference pins on the working table to keep the block in position after turning it upside-down: on the machine the zero point then needs to be positioned exactly in the middle of these pins.
- **None** means that no translation is applied. The zero-position of the CAD data will correspond with the work-piece zero-point on the machine.
- **Custom** makes it possible to use any own defined translation, to be entered in the edit boxes. The value that you enter is the translation between CAD coordinates and workpiece coordinates. The edit boxes also show the actual translation for any of the other options.

Z:
- **Make top of block zero** has just been explained: Z=0.0 will be on the top surface of the block. So the CAD data will be translated in a way that all points have negative Z-values. This option is the most convenient one because it will be easy to set the zero-position on the machine.
- **Make center of block zero**: the workpiece zero point will then be exactly in the middle of the block for Z.
- **Make bottom of block zero** will put Z=0.0 at the bottom of the block. So the CAD data will be translated in a way that all points have positive Z-values. Advantage of this option is that the workpiece zero point can be the same for any block thickness. Note that in case of ballnose cutter the tip of the cutter can still travel below Z=0, see the Ambient page.
- **None** means that no translation is applied. The zero-position of the CAD-geometry will correspond with the workpiece zero point on the machine.

- **Custom** makes it possible to use any own defined translation, to be entered in the edit box. The value that you enter is again the translation between CAD coordinates and workpiece coordinates.

For Rotation axis machining only the X and Z coordinates can be translated. The Y coordinate value can not be translated as the tool does not move along the Y axis during rotation axis machining. In this case also some of the Z-options are not available either as they would not make sense.



When you have checked the option **Use rotation axis tilt option** on the General tab of the Part parameters, a different set of Translation options will be available. See the illustration above.

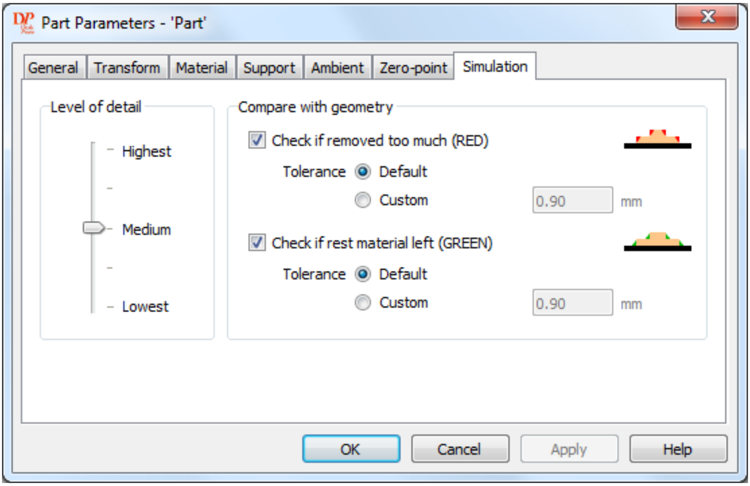
As explained for the Tilt option settings (rotation axis tilt option) of the Operation parameters, the X-translation will determine the location of the part when the tilt angle has been applied. The workpiece zero point (X=0.0) needs to be exactly on this 5th axis (so on the tilting rotation point). The small illustration on the dialog shows that a larger distance between the axis and the part will result in a larger Z-displacement when rotating. So the **Distance to material block** must be carefully set in order to have the toolpaths aligned for operations with and without a tilt angle.

The **Resulting Translation** will be different from the Distance to material block, as the zero point of the STL file in most cases is not located on the left edge of the block.

A Y-axis translation is not possible here: this tilt option is combined with rotation axis (4th axis) toolpaths, where Y-translation is not applicable.

Simulation parameters

(Vector, Geometry and Bitmap Settings)



A **Simulation** is a drawing on screen that shows you what the resulting machined part will look like. This can be used to check things like the resulting surface smoothness, error movements that would damage the part, rest-material where the cutter cannot reach, etc. In this dialog you can set the simulation parameters. You can read how to display the simulation on the Simulated Operations dialog page.

The **Level of detail** sets the accuracy of the simulation to be calculated. The simulation is in fact a Z-grid like used in many other DeskProto calculations: the level of detail sets the number of grid cells used. The higher the level of detail, the longer it will take to calculate the simulation and to draw it.

Five preset options are available to set the number of cells in the simulation grid:

Lowest 200

Low	500
Medium	1000
High	2000
Highest	5000

This number of cells is used for the longest side of the block to be simulated. The number of grid cells along the short side of the block is calculated proportionally.
In most cases the default Medium setting will be a good choice. A higher level of detail will be needed only when you want to zoom in onto some detail. This will make both calculation and display of the simulation much slower though.

Compare with Geometry offers the possibility to check the difference between the resulting part and the original STL file geometry (only in the Geometry settings). DeskProto will calculate the distance between the simulation and the STL geometry, and will apply a color on the simulation when this distance is above a certain tolerance value.

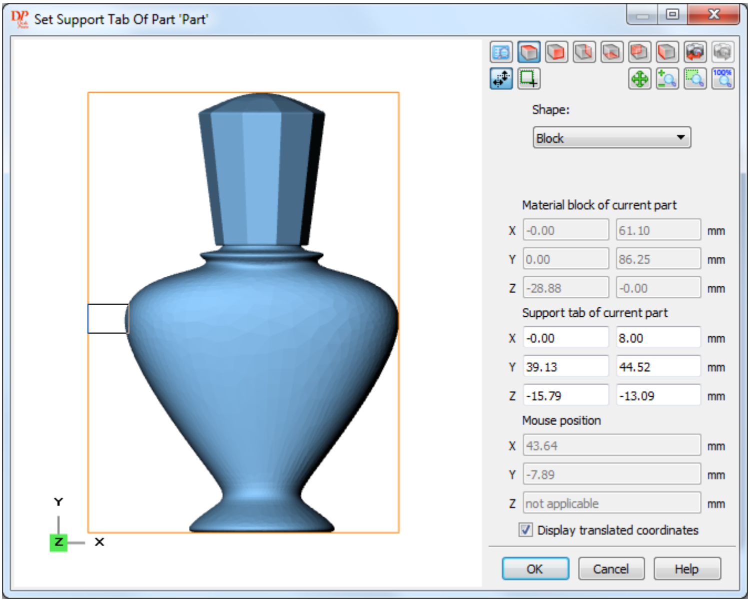
Check if removed too much (RED) will show it in case DeskProto has removed too much material. Note that the red will only be visible when you have turned off the rendered geometry in the Items visible dialog, otherwise the geometry will hide the red.

Check if rest material left (GREEN) will show in case too little material has been removed because the cutter could not reach a certain position. This may happen on many occasions, like:
- toolpath distance too large (and a ballnose cutter)
- small hole where the cutter does not fit inside
- sharp inner corner, which will be machined with the Radius of the cutter

A **Tolerance** value decides whether or not to apply such color red or green. As a default this tolerance has been set to 0.1% of the material block size, so the largest dimension of the block divided by 1000. When you set it to Custom you can enter any value that you need in the edit box.

Note: DeskProto does not support simulation of rotation axis toolpaths, so when on tab page General option Use rotation axis has been checked the simulation tab page will not be displayed.

4.3.3 Set Support Tab



The Set support tab dialog makes it possible to set the size and position of one geometry support tab. You can reach this dialog by using an Edit button on the Support tab of the Part parameter dialog.

The dialog shows a new drawing of your part, with a rectangle that indicates the support tab that you are editing. You can edit the tab by 'picking' one of the sides of this rectangle with your mouse and then moving it (as the button 'Adjust boundary' now is active).

Using the buttons on the right you can change the drawing. Twelve of these are standard DeskProto buttons to set the viewpoint, of which the use is known and needs no explanation here. Note that mouse rotation is not present: this dialog only uses the six main views.

The first button on the first row shows the **Items visible** icon that is also used in the main window. This icon however opens a special version: the 'Items visible for Set Graphically' dialog. It will be clear that here you can select the items to be shown in the drawing.

The first two buttons on the second row call the main function of this dialog: graphically set the support tab. Note that these are “mouse buttons” as well: of the five mouse buttons only one is active at any time.

This button sets the mouse function to **Resize the current tab** by dragging each of the four sides to a new position.

This button sets the mouse function to drag a complete new rectangle to define a **New support tab**.

Below the buttons you can use the combo-box to select the **Shape** of the support tab.
By default five shapes are available: Block, Cone, Cylinder, Pyramid and Wedge.
For the Cone, Pyramid and Wedge the orientation of the support is determined by which support tab you are editing: left, right, front or back.

You can also set the new support tab by entering minimum and/or maximum values for X, Y and Z in the six edit boxes. The dimensions of the current material block are shown to assist you, so are the coordinate values of the current mouse position.

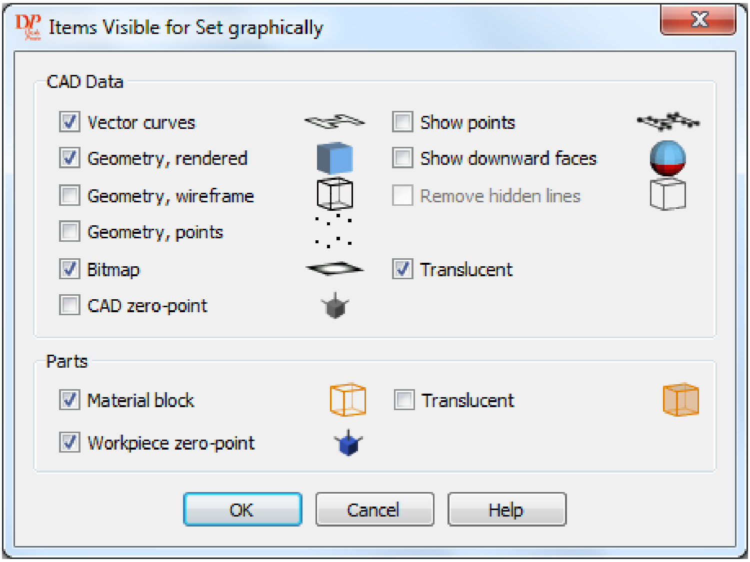
As a support tab is aligned with the main axes, this resizing only makes sense in the six main views (looking along X, Y or Z): other viewpoint positions are not possible in this dialog. For each of the main views of course only two coordinates can be changed: to change the third as well you have to select a different main view using one of the six view buttons.

Note:
It is possible to add your own support shape.
The base support tabs are present as STL files in files in C:\Program Files\DeskProto 7.0\Supports\
You can add any STL file here, and use this geometry as a support tab. For instance a special shape that matches a fixture that you have made.
Important is that the geometry in this STL file must be sized to exactly fit inside a cube of 1 by 1 by 1 mm: only then the dimensions that you will set in this dialog will be set correctly (internally DeskProto creates the support by copying, scaling and rotating the base support tab as defined in this file).

For software specialists:
you can set the location of this supports directory in the registry, at

HKEY_CURRENT_USER\Software\Delft Spline Systems\DeskProto\7.0\Preferences\File Locations\SupportsLocation
Do not change this unless you are sure what you are doing.

4.3.4 Items Visible for Set Graphically

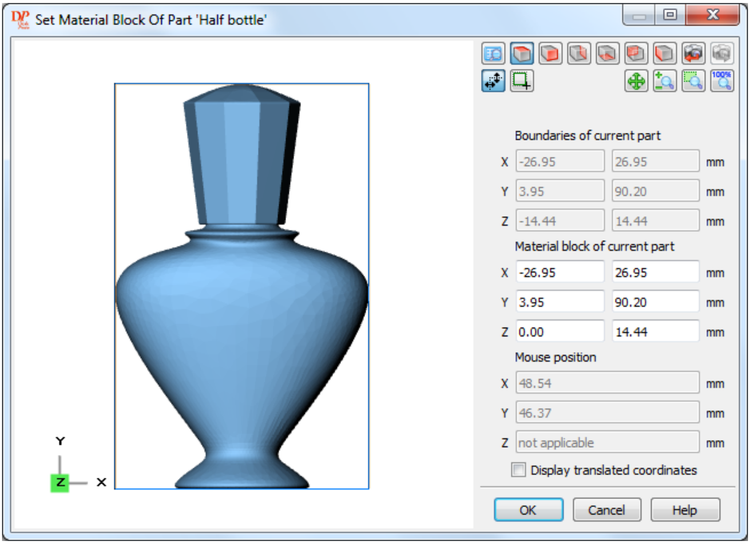


In DeskProto a number of dialogs are present that show a drawing of the part in order to **graphically** set a parameter:
Set Material block (Part parameters)
Set Support tab (Part parameters)
Set Center (Operation parameters, for strategies Circular and Radial)
Set Area (Operation parameters)
Set Freeform area (Operation parameters).

When graphically setting this parameter it may be handy change which items are visible in the drawing. This dialog allows you to switch each of the available items on and off.

For more information see the Help page of the Items Visible dialog for the main screen.

4.3.5 Set Material Block / Area



This dialog is used for two settings in DeskProto, each with a different name. The contents of the dialog is identical though.
The dialog **Set Material Block** makes it possible to graphically set the size and position of the material block. You can reach this dialog by using the button Set Graphically on the Material tab of the Part parameters dialog.
The dialog **Set Area** makes it possible to graphically set the size and position of the area to be machined. You can reach this dialog by using the button Set Graphically on the Area tab of the Operation Parameters dialog.

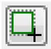
The dialog shows a new drawing of your part, with a rectangle that indicates the block or area that you are editing. You can edit the tab by 'picking' one of the sides of this rectangle with your mouse and then moving it (as the button 'Adjust boundary' now is active).

Using the buttons on the right you can change the drawing. Twelve of these are standard DeskProto buttons to set the viewpoint and the mouse function, of which the use is known and needs no explanation here. Note that mouse rotation is not present: this dialog only uses the six main views.

The first button on the first row shows the **Items visible** icon that is also used in the main window. This icon however opens a special version: the 'Items visible for Set Graphically' dialog. It will be clear that here you can select the items to be shown in the drawing.

The first two buttons on the second row call the main function of this dialog: graphically set the block or area. Note that these are “mouse buttons” as well: of the five mouse buttons only one is active at any time.

This button sets the mouse function to **Resize** the current block or area by dragging each of the four sides to a new position.



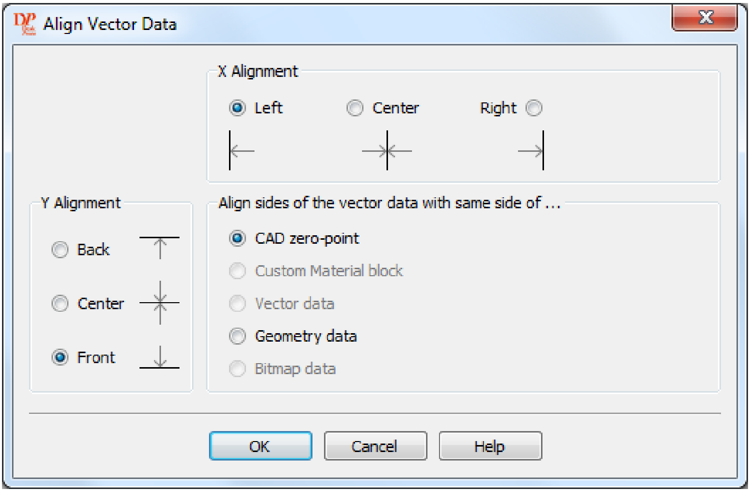
This button sets the mouse function to drag a complete new rectangle to define a **New** block or area.

You can also set the new block or area by entering minimum and/or maximum values for X, Y and Z in the six edit boxes. The boundaries of the current part are shown to assist you, as are the coordinate values of the current mouse position.

When you change the block or area, you will see that both the old and the new block remain visible: the changes will become effective after pressing the OK button.

As a block/area is rectangular and aligned with the main axes, this resizing only makes sense in the six main views (looking along X, Y or Z): other viewpoint positions are not possible in this dialog. For each of the main views of course only two coordinates can be changed: to change the third as well you have to select a different main view using one of the six view buttons.

4.3.6 Panning Alignment



This dialog is used both for the Vector settings (Align Vector Data) and for the Bitmap Settings (Align Bitmap Data), and can be reached using the "Align to..." button in the Part parameters: for Vector tab Transform, for Bitmap tab XY Transform.

When you use more than one type of CAD data for one part, it may be needed to align these data types with one another. This dialog makes such alignment easier.
Only alignment by panning along X and Y is supported, panning Z and rotation are not possible here.

The thing that will be aligned is the **Bounding box** of the Vector curves respectively of the Bitmap image.

This bounding box can be aligned with one of five entities:

- **CAD zero point**. Always possible.
- **Material block**. Only available in case a custom block has been defined.
- **Vector data**. The bounding box of all vector curves. Only available for bitmap data, in case one or more vector files have been loaded.
- **Geometry data**. The bounding box of all geometry. Only available in case one or more geometry files have been loaded.
- **Bitmap data**. The sides of the bitmap image. Only available for vector data, in case a bitmap has been loaded..

For **X-Translation** three options are present:

- **Left** means that the Left side of the Bounding box will be aligned with the left side of the entity.
- **Center** means that the Center of the Bounding box will be aligned with the center of the entity.
- **Right** means that the Right side of the Bounding box will be aligned with the right side of the entity.

The CAD zero-point does not have a side, so here the Bounding box will be aligned with the point.

For **Y-Translation** the options are called Back, Center and Front, but are in fact the same as for X.

4.3.7 Vector Operation Parameters

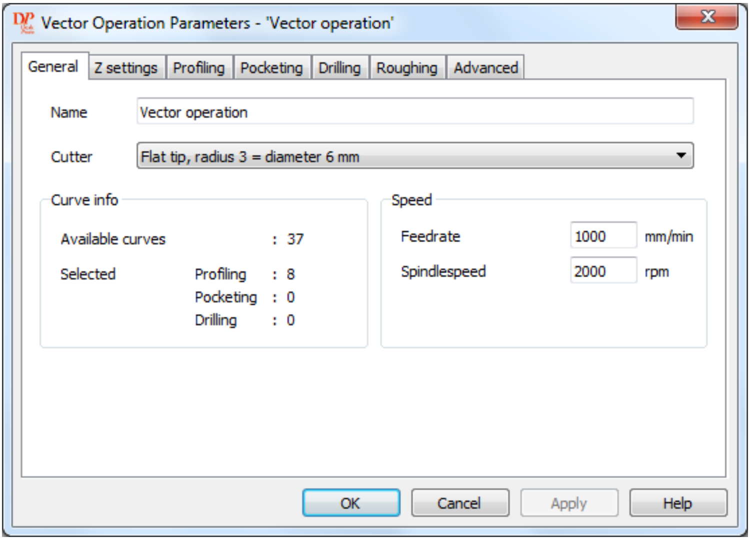
DeskProto features three different types of Operations: this dialog is for the Vector Operation -- in addition also dialogs for a Geometry Operation and for a Bitmap Operation are available.

The Vector Operation parameters are divided into 7 sections by tab pages. In the Free edition of DeskProto only the first three tab page are available, offering less parameters than the Expert and Multi-Axis editions. In the Entry edition only the Advanced tab is missing.

This dialog can be reached via the Parameters menu, third option.
Or you can double-click on an operation-item in the project tree (one of the third level items).
Or right-click on an operation-item and select Operation Parameters in the context-menu.

This same dialog is used for the Default Vector Operation parameters, only with an extra button **Restore DeskProto defaults** to reset the original default parameters.

General parameters



Name

The name of the operation can be changed; use a meaningful name to easily remember the purpose of each specific operation. The name is meant for your convenience only, it is not used in the NC program file. It will be used for the file name though in case the NC output is in more than one file.

Cutter

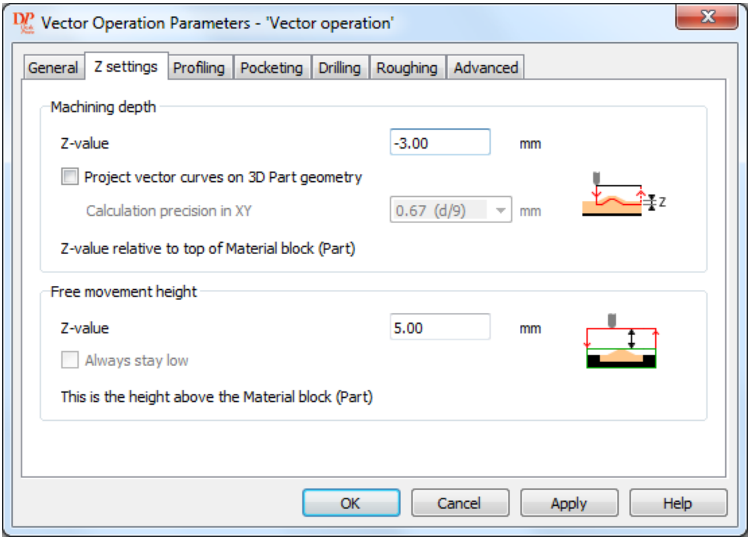
You can select a cutter from the cutting tool library using the small arrow button at the right. Adding new cutters to the library, changing an existing cutter or just retrieving information on a cutter can be done using Library of cutters (Options menu).

Curve info

The Vector data that has been loaded consist of a number of vector curves. In order to use curves for toolpath calculation they need to be selected first: for Profiling, for Pocketing and/or for Drilling. It is possible to combine two or three toolpaths types in one Operation. In case no curves have been selected the Vector operation is invalid as then no toolpaths can be calculated. Selecting curves cannot be done on this tab page, only on the three tab pages just mentioned.

The **Speed** settings are identical to the settings offered in the Geometry Operation Parameters: see the explanation on **Feedrate** and **Spindle Speed** there.

Z settings parameters



The **Machining depth** is the Z-coordinate for the actual toolpath. You can describe this as the *pen-down level* when plotting a 2D vector file. The depth is measured from the top of the block. This means that a positive value does not make sense here.
In case of a 3D polyline the machining depth is taken relative to the Z-value of the curve.

The option **Project vector curves on 3D part geometry** (not available in the Free and the Entry edition of DeskProto) is ideal for instance to engrave a logo or text onto a 3D design: the 2D vector curves will be converted into 3D toolpaths. When you check this option, the Machining depth is no longer interpreted as a standard Z-level, however is taken relatively to the Z-level of the geometry at that point. So a level of –1 mm results in a groove of that depth over the 3D part.
The projection is vertical, meaning that on curved surfaces the 2D drawing will be distorted: a circle that is projected on a sloped surface will be changed into a ellipse.

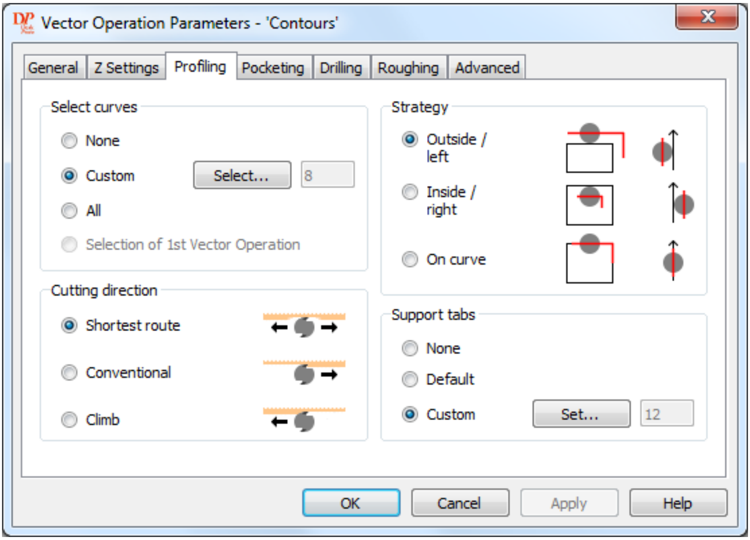
For this projection DeskProto will take into account the shape of the cutter in 3D.
This option is only available when geometry has been loaded, and when at least one Geometry Operation is present before this Vector operation. This Geometry Operation is needed to make sure that all material above the geometry has been removed.
Projection is not possible when in the Project parameters the option "Use Z-values" has been checked, as then the Z-values are taken relative to the Z-value of the vector curve.

In order to calculate such projected vector toolpath DeskProto needs to make a Z-grid. The option **Calculation Precision in XY** allows you to set the precision (the cell-size) of this Z-grid. The smaller the value entered, the more accurate the toolpath will be, and the more calculation time needed. This Z-grid will be calculated only for the area covered by the vector file.

The **Free movement height** specifies the height at which the cutter can freely move over the block, so without touching material. As in the analogy with a pen plotter the machining level is the *pen-down level*, the free movement height can be seen as the *pen-up level*. This height is used to move the cutter from one position to another without machining (positioning movements). It is set as the height above the top of the block, so negative values are permitted here. The positioning movements will be done in Rapid mode (so at the maximum speed of the machine).
More information on positioning movements on the Movement tab of the Geometry operation.

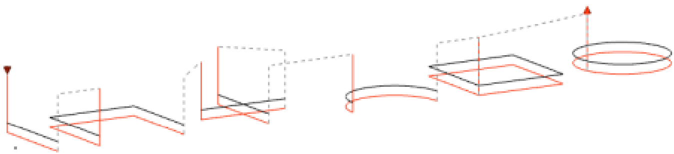
When the option **Always stay low** is checked, the positioning moves are done at normal feedrate. When only Vector data is present the Z-level of these positioning movements will not change: the specified height above the top of the block. When a Geometry has been loaded as well the positioning movement may be done below the top of the block: each movement then is done at the Free movement height above the highest point of the geometry over which the cutter moves.
Always stay low can not be checked when Roughing layers are used.
For very small positioning movements DeskProto will not let the cutter rise to Free Movement level, see the explanation on the Geometry Operation page.

Profiling parameters



Three different toolpath types can be generated based on the vector curves: for Profiling, for Pocketing and for Drilling. This tab page contains the settings for **Profiling**: the then cutter will exactly follow each curve that is selected.

Select curves allows you to specify which of the available vector curves need to be used to generate profiling toolpaths. The option **None** will be clear. To make a **Custom** selection it is needed to open dialog Edit curve selection using button **Select...**.
The fourth option, to use the **Selection of 1st Vector Operation**, of course is available only in case a first vector operation with such selection is present.



For Profiling it is not possible to select single point 'curves'. The drawing above contains 8 vector curves: one single point (on the left side left), five open curves (the + consist of two lines) and two closed curves (on the right side). Selecting **All** for this drawing will result in 7 curves being selected here: all except for the single point.

Three **Strategy** options are present to follow the curve:

Outside/left is meant to produce parts that exactly match the shape of each curve: the profile of the part. So the cutter will move along a path that follows the curve at exactly R mm distance (where R is the radius of the cutter), on the outside of the curve. Outside of course can be defined only for closed curves, for an open curve the cutter will travel on the left side of the curve (as defined by the curve direction in the vector file).
For closed curves inside this curve (so nested curves) the cutter will travel on the inside of the curve (holes in the part). The next nesting level again will be with toolpath on the outside, and so on. DeskProto will start with the innermost curves and than works it's way to the outside, in order to keep the part clamped when machining.

Inside/right is meant to produce holes that exactly match the shape of each curve: the profile of the hole. So the cutter will move along a path that follows the curve at exactly R mm distance (where R is the radius of the cutter), on the inside of the curve. Inside of course can be defined only for closed curves, for an open curve the cutter will travel on the right side of the curve (as defined by the curve direction in the vector file).
For closed curves inside this curve (so nested curves) the cutter will travel on the outside of the curve (islands within the hole). The next nesting level again will be with toolpath on the inside, and so on. DeskProto will start with the innermost curves and than works it's way to the outside, in order to keep the part clamped when machining.

Strategy **On curve** means that the center of the cutter will exactly follow each curve that was selected. The resulting shape(s) then will depend on the diameter of the cutter that is used. Now DeskProto really acts like a plotter. The cutter will exactly follow the lines as defined in the 2D vector file, at a certain machining depth (**pen-down level**). For positioning moves in-between the cutter rises to Z-free height (**pen-up level**).

All three strategies can also be used for vector curves that contain Z-values! That is ideal for instance for cutting holes in thermoformed parts. The curve then will copy the Z-value of the point in the curve.
For strategies Outside/left and Inside/right this will be the closest point of the curve. Note that when toolpaths at different heights share the same XY position the algorithm now may copy an incorrect Z-value, so please check your toolpaths when using this option.

Cutting direction

For strategies Outside/left and Inside/right it is possible to choose between **Conventional**, **Climb** and **Shortest route**. The small drawings explain the difference between these three options: it is about the relation between the direction of milling and the rotation direction of the tool (normally clockwise). The choice made here will affect the surface quality of the machined profile, the best choice is different for each material. The option Shortest route will use the direction to make the positioning movements from curve to curve as short as possible.
For strategy On Curve it is not possible to define Conventional or Climb, for that strategy you can select either **Original** (as defined in the CAD file), **Reversed** or **Shortest route**. In the latter case DeskProto again will optimize to reduce the total distance for the positioning movements.

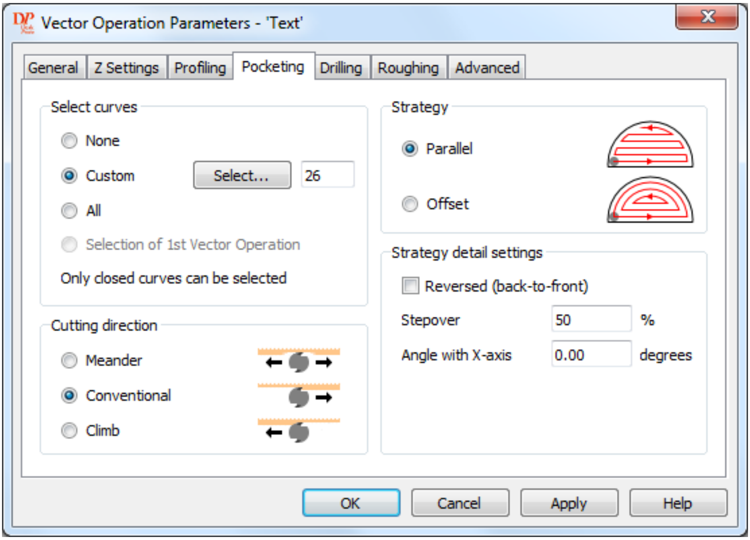
Support tabs

Support tabs can be used to keep the part connected to the rest of the block. When cutting a closed profile at full depth in sheet material a separated part will result: the material inside the profile is no longer is connected to the rest of the sheet. Support tabs are *interruptions in the toolpath*, acting as connection bridges to hold the part on its place. this allows you to clamp the materials on its corners and machine your parts, each on a different position, without the need to clamp each separate part.

This tab page is about Vector support tabs, made by changing the machining depth in a 2D toolpath. Not to be confused with Geometry support tabs, made by adding small blocks to the geometry.

Three options are present:
None will be clear: no support tabs
Default will automatically generate supports conform the Default profiling tabs settings.
Custom allows you to exactly define the location and size of each tab, using the Set Profiling tabs dialog.

Pocketing parameters



Three different toolpath types can be generated based on the vector curves: for Profiling, for Pocketing and for Drilling. This tab page contains the settings for **Pocketing**: the cutter will remove all material within a closed curve. The resulting hole in the block of material is called a pocket: hence the name pocketing.

Select curves allows you to specify which of the available vector curves need to be used to generate pocketing toolpaths. The option **None** will be clear. To make a **Custom** selection it is needed to open dialog <u>Edit curve selection</u> using button **Select...**.
The fourth option, to use the **Selection of 1st Vector Operation**, of course is available only in case a first vector operation with such selection is present.

For Pocketing **only closed curves can be selected**, as an open curve does not define a pocket. The drawing above contains 8 vector curves: one single point (on the left side left), five open curves (the + consist of two lines) and two closed curves (on the right side). Selecting **All** for this drawing will result in 2 curves being selected here: only the two closed curves.

Two **Strategy** options are present to create a pocket:
The **Parallel** strategy fills the pocket with toolpaths parallel to the X-axis.
The **Offset** strategy fills the pocket with toolpaths that follow the shape of the outside curve (the contour that defines the pocket).
Each strategy is clearly illustrated by the small icon drawing in the dialog.

Each of these two strategies has its own set of **Strategy detail settings**.

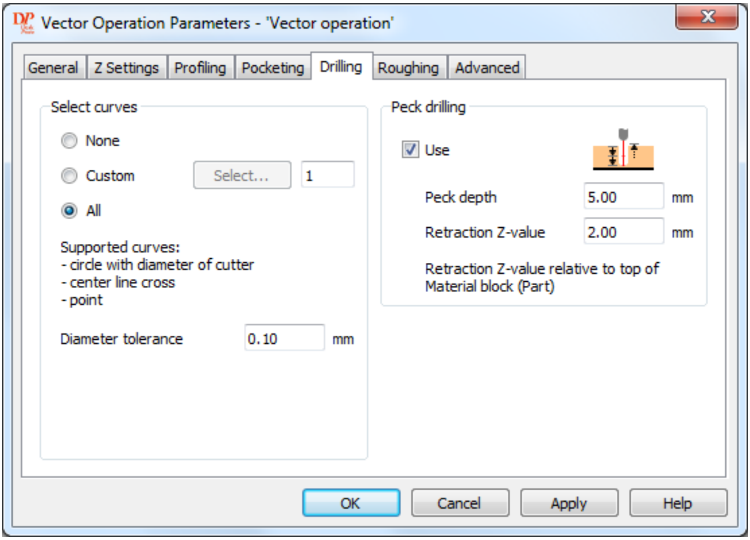
For **Parallel**:
The parallel toolpaths normally start minimum Y and continue to maximum Y, **Reversed** will reverse that direction.
Stepover is the distance between two parallel toolpaths. It is set as percentage of the cutter diameter. So for a cutter of 10 mm diameter, a Stepover of 50% will mean a toolpath distance of 5 mm, and 80% will mean 8 mm.
By default the pocketing toolpaths as said are parallel to the X-axis, the option **Angle with X-axis** allows to change this.

For **Offset** only two detail settings are present as the Angle does not apply:
Offset toolpaths normally start with the outmost path and then progress to the middle. **Reversed** means to start in the middle and progress to the outmost path.
The **Stepover** setting is the same for both strategies, set as percentage of the cutter diameter.

The **Cutting direction** applies to the toolpaths when emptying the pocket. So for Offset toolpaths outside-in (that start with the outmost path and proceed to the center of the pocket) the milling direction along the outer profile of the pocket will be the reverse.

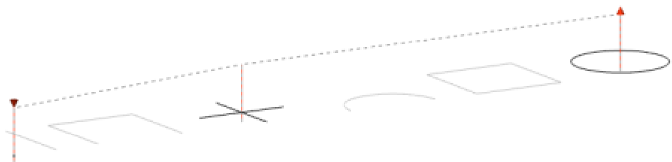
For both strategies it is possible to add a toolpath that follows the contour of the pocket. This can be done on the Roughing tab: after setting an *Allowance* such *Profile path* can be added in order to remove the allowance and make the pocket smooth.

Drilling parameters



Three different toolpath types can be generated based on the vector curves: for Profiling, for Pocketing and for Drilling. This tab page contains the settings for **Drilling**: the cutter will move along the Z-axis and drill a hole in the block, at the XY location as defined in the drawing, and a machining depth as defined on tab Z Settings.

Select curves allows you to specify which of the available vector curves need to be used to generate pocketing toolpaths. The option **None** will be clear. To make a **Custom** selection it is needed to open dialog Edit curve selection using button **Select...**.



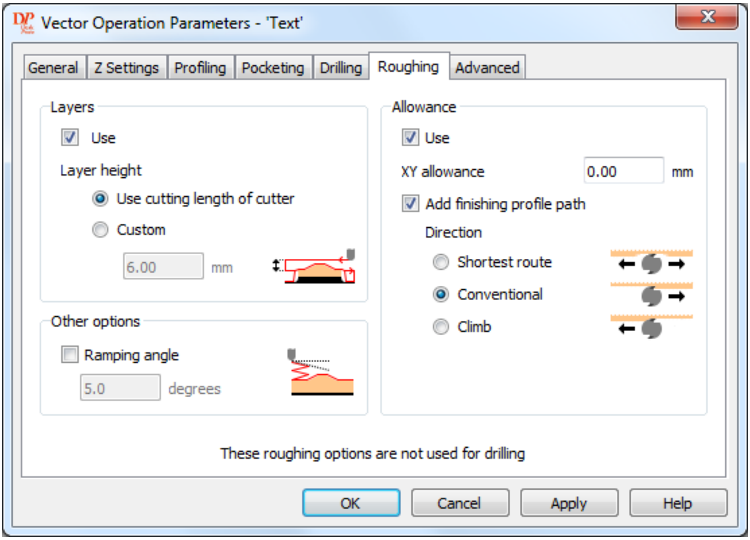
For Drilling three types of curves can be selected:
- single points (defining the XY position of the center of the hole)
- + signs (so two single lines that together form a + sign), where the length of each line equals the cutter diameter.
- circles with a diameter that equals the cutter diameter

The drawing above contains 8 vector curves: one single point (on the left side left), five open curves (the + consist of two lines) and two closed curves (on the right side). Selecting **All** for this drawing will result in 3 curves being selected here, as the diameter of the cutter is correct for both the + and for the circle. For a cutter with any other diameter only one curve will then be selected: the single point.

Setting a **Diameter tolerance** may be needed to make the above selection process work correctly. For instance a circle of 4.0 mm diameter in an inch drawing may have been rounded to 0.1575 inch. When DeskProto uses the exact diameter (0.1574803 inch) it will not find that circle. Clearly you intended to use it, which is solved by applying this tolerance.

For most toolpaths the chips that are cut off the block can freely fly away. Not so for drilling, as then the cutter is surrounded by solid material on all sides. This means that the grooves in the cutter at some point will be completely filled with chips, which will soon be compressed to one almost solid mass that will make cutting impossible. The solution is to regularly retract the cutter to a position above the block, thus freeing the chips. This process is called **Peck drilling**, as it resembles what a woodpecker does. This parameter offers two sub-settings:
Peck depth is the Z-distance after which a (next) retraction movement needs to be done
The **Retraction Z-value** sets the height (above the top of the block) that will be reached in the retraction movement.

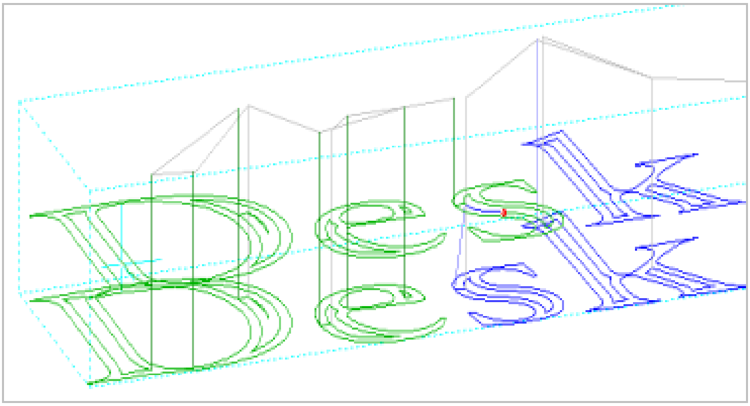
Roughing parameters



Roughing is applied when the cutter cannot reach the required Machining depth in one cutting movement, either because the cutter's cutting length is not sufficient or because the material is too hard to machine all at once. The total depth then will be reached in a number of cutting movements (layers), each on a lower Z-level.
Roughing can also be used to achieve a high quality result: when the roughing toolpaths have removed most of the material, the load for the finishing toolpaths will be low which sill reflect on the resulting quality.

When you un-check **Use Layers** the cutter will immediately go down to full depth. This choice is not allowed for the first Vector operation of a part: DeskProto does not allow the cutter to remove more material than possible for the cutting length of the cutter. In case you are sure that sufficient material has already been removed to allow skipping the layers (for instance by a Geometry Operation) you can add an extra Vector operation to the part, before the current operation, and make it invisible (an unused dummy operation).

The **Layer height** that is specified determines how deep the cutter may go into the full material. You can set this Layer height to the Cutting length of the cutter or to a Custom value. The custom value may not be higher than this cutting length though. The default layer height equals the whole Cutting length of the cutter. In most cases it is preferable to use a smaller **Custom** layer height, as with a tough material you do not want the cutter to use its total cutting length. As a rule of thumb: set the Layer height equal to the cutter's (flute) thickness.



For Geometry toolpaths and Bitmap toolpaths the roughing layers are completely machined, one by one, from top to bottom.
For Vector toolpaths the Roughing Layers are machined in a different way: the layers are machined per separate curve, of course again from top to bottom. Only "nested curves" are treated as one group: the complete "nest" is done layer by layer. Like in the image above, where this sequence is being followed: layer 1 for the D nest, layer 2 for the D, layer 1 for the E nest, layer 2 for the E, currently machining layer 1 for the S nest (the red dot shows the current cutter position), and so on.

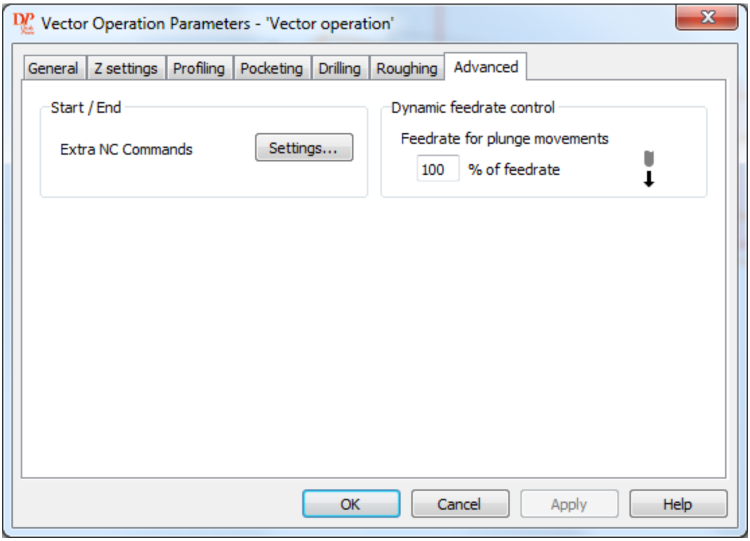
The **Ramping angle** is used when starting to machine. The cutter will then first move to the correct XY position: exactly above the first point to be machined, on the Free movement height Z-level. Normally the cutter then will move down to that first point in one vertical downward movement (plunge).
Such vertical downward movement is not ideal: many cutters do not like that, and the chips cannot escape from the deep round hole that is created. Entering a Ramping angle makes DeskProto replace this vertical movement by a series of ramping movements: go down along a sloping line. See the small picture in the dialog for this option. You can set how steep this sloping line needs to be by entering an angle value in degrees: this is the angle between the sloping line and a horizontal line.
Important: Ramping uses the **space that is available above the first movement** (a straight line). In case that first toolpath is too short then ramping is not possible.

Setting an **Allowance** keeps a cutter at that distance of the actual curve when roughing. The allowance prevents that the cutter takes away too much material (this can happen as the cutter will vibrate and may bend when roughing). The allowance also improves the resulting surface quality, as during finishing the tool will remove the same small amount of material all the time. It is the equivalent of the Skin in a Geometry operation, however applied only to X and Y.

When an Allowance is set it is possible to **Add a finishing profile path** to the operation. This path will follow the exact curve and remove the allowance, using the same cutter. In order to finish with a different cutter (for instance smaller in case of small details) you need to use separate operations for rouging and finishing.

For this finishing profile path you can finally define the **Direction** of milling: **Shortest route**, **Conventional** or **Climb**, see the explanation above (tabs Profiling and Pocketing).

Advanced parameters



Start / End offers the option to add extra commands to the NC program before the operation toolpath starts and/or after it has ended. These commands can be movement commands and/or user defined commands. They can be used for instance to make the cutter move to a safe position before rotating the 4th axis. Button **Settings...** will open the Operation Start/End settings dialog.

Using the **Feedrate for plunge-movements** it is possible to decrease the feedrate when the cutter moves downwards. This may be needed when machining in metals, as fast plunge movements may damage the cutter (many cutters have problems with drilling). It is expressed as a percentage of the normal feedrate for this operation.
The rate you enter here will be used for movements that go down along an angle that is more than (steeper than) 30 degrees.
For downward movements along an angle less than 30 degrees the reduction will be smaller: DeskProto will apply the rate as specified + 20. So when you have set the plunge rate to 40 %, these movements will be reduced to 60 % of the normal feedrate.

4.3.8 Geometry Operation Parameters

DeskProto features three different types of Operations: this dialog is for the Geometry Operation -- in addition also dialogs for a Vector Operation and for a Bitmap Operation are available.

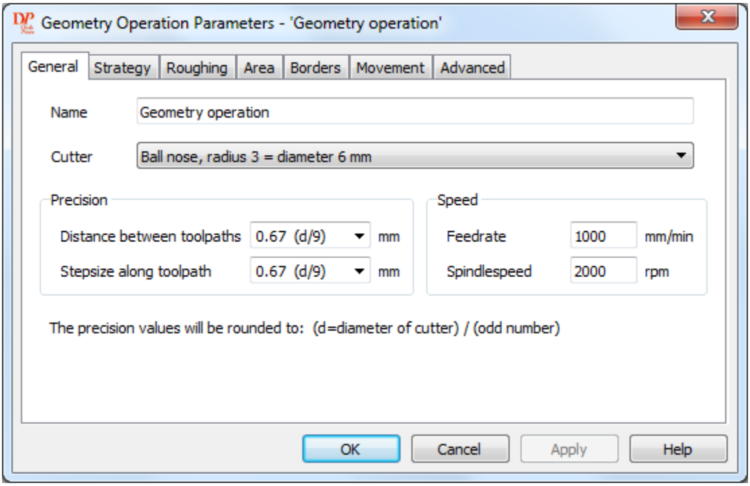
The Geometry Operation parameters are divided into 7 sections by tab pages. The further to the right, the more advanced the parameters in the tab. In the Free edition of DeskProto only the first tab page is available and In the Entry edition only the first and the third tab page, offering less parameters than the Expert and Multi-Axis editions.

This dialog can be reached via the Parameters menu, third option.
Or you can double-click on an operation-item in the project tree (one of the third level items).
Or right-click on an operation-item and select Operation Parameters in the context-menu.

This same dialog is used for the Default Geometry Operation parameters, only with an extra button **Restore DeskProto defaults** to reset the original default parameters.

The Geometry Operation parameters are identical to the Bitmap Operation parameters. The Geometry Operation applies them on the Geometry, the Bitmap Operation applies them on the Bitmap relief.

General parameters



Name

The name of the operation can be changed; use a meaningful name to easily remember the purpose of each specific operation. The name is meant for your convenience only, it is not used in the NC program file. It may be used for the file name though, in case the NC output is in more than one file.

Cutter

You can select the Cutter to be used from the a list (that will open when you click on the currently selected cutter). Which cutter is best depends on how the geometry is shaped. Generally speaking:
- For freeform surfaces use ballnose cutters to reduce the staircase effect.
- For 'straight/square' geometries use a flat cutter to get flat horizontal surfaces and sharp inner corners.

The larger the cutter, the smoother the resulting surface and the faster the machining. Use small cutters only in case of small details. You can also use a large cutter for the complete part and a small one later for some detailed areas. Adding or removing cutters to/from the list can be done in the Cutter library (Options menu), where you can also Edit existing cutter definitions.

Precision
Here you can enter the accuracy to be applied. Two parameters are present: **Distance between toolpaths** (also called the Stepover) and **Stepsize along toolpath** (each path is built as a large series of movements, each step being a very small straight line). Normally both distances are set equal. The smaller the distances, the more accurate the model, however also proportionally more time will be needed for both calculation and milling.
In case you have selected to use the rotation axis, one of both precision values should of course be an Angle in degrees. Still, as this is easier to imagine, a distance in mm or inch is used. DeskProto will convert this value to degrees at the outside of the (cylindrical) area to be machined.

An example will illustrate why this is called Precision: in case the distance between the toolpaths is set to 1 mm and a cube of 10.5 mm has to be machined, then this is not possible as the resulting cube model will be either 10 or 11 mm (DeskProto will in fact make it 11). This inaccuracy is a drawback of the algorithm that DeskProto uses, and is more than compensated for by its advantages such as calculation speed, robustness, ability to work with incomplete/incorrect geometries, and ease of use. In fact DeskProto has been designed for prototyping, not for production tooling.

The precision values used will be rounded to a value that is calculated by dividing the diameter of the cutter by an odd number. The 8 predefined values offered by DeskProto do match this formula. You may enter a custom value as well, which in case it does not match will be rounded to a more precise value that does match the formula. The reason to use this rounding is that the resulting (physical) part will be more accurate, due to the algorithm used by DeskProto for its calculations. For example: When the diameter of the cutter (d) is 4.0, and you enter a precision value of 1.0, it will be changed to 0.8 (d/5 = 0.8). The value of 1.0 that you have entered will nevertheless be saved in the project. So when you later change the cutter, the precision will still be (close to) 1.0. For advanced users some detail precision settings ("subsampling") are available on the Strategy tab page. These will enable you to use a precision that is higher than the toolpath distance.

Note 1:
In this calculation the **flute diameter** of the cutter is used (so not the shaft and not the tip).
Note 2:
Be careful with a large Stepsize. The cutter will move in a straight line to the next calculated position, so a large step might damage some in-between geometry. This is most likely to happen in case of vertical walls, and can be corrected by the option Protect vertical surfaces.

Note 3:
When a **Skin** is applied in the Roughing parameters, the diameter of the resulting virtual cutter is used. Say you use a ballnose cutter with a 6 mm diameter, and a skin of 0.5 mm. Then DeskProto will do the calculations with a virtual cutter of 7 mm diameter (radius 3.0 + skin 0.5 = 3.5). This means that the first option in the combo-boxes (drop down menu) for Precision now will be "7.0 (d/1)" As this is larger than the diameter of the actual cutter, this first option will be grayed out.
Same for other cutter types, though then the geometry of the virtual cutter will be different (sharp corners will be rounded).
For negative skin value DeskProto allows precision values that are larger than the virtual cutter.

Speed
The **Feedrate** is the speed with which the cutter moves through your material. The value you enter here must be between the minimum and maximum feedrate values permitted for the machine you selected for the project.
Distinguish the feedrate from the actual cutting speed of the tool's cutting edge, which is determined by the rotation speed (spindle speed) and the diameter of the cutter.
The units used for the Feedrate are set in the postprocessor of the machine that you selected for your part. DeskProto does not check or even understand these units: it just copies the number that you enter here to the NC file. Whether or not you can use decimal values for the Feedrate also depends on the postprocessor settings: on the Feedrate page you can define whether or not decimals are used.

Generally speaking a tougher material will require a lower feedrate. Same for a smaller cutter. A very handy option is to let DeskProto automatically reduce the Feedrate in high Chipload conditions, on Tab page Movement of this dialog.

The **Spindle speed** is the rotation speed of the cutter. The unit is *rpm*, which stands for rotations per minute. The value you enter here must be between the minimum and maximum spindle speed values permitted for the machine you selected for the project. The smaller the diameter of the tool, the higher the spindle speed needed in order to get the same actual cutting speed.

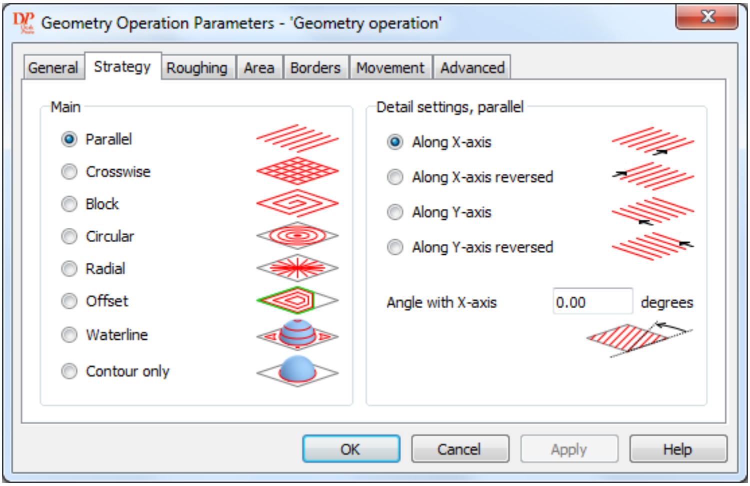
Note:
It is possible to check **Automatic Speed setting** for a cutter in the Cutter definition. In that case when the cutter is chosen the Feedrate and Spindle

Speed will be set automatically. That is quite useful for very thin cutters that need a low feedrate and a high spindle speed as otherwise you could forget to set the correct speeds and break your cutter.

Note:

Each Speed option is only available in case your machine supports it, if not the option will be grayed out in this dialog. For instance on many machines the Spindle speed cannot be set from the computer but only using a knob on the machine. This setting (so whether or not this is possible on your machine) can be made in the Postprocessor dialog (Options menu).

Strategy parameters

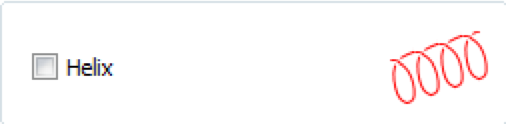


The eight available **Main** strategies should be clear from the pictures drawn in the dialog. Each main strategy (at the left) has its own **Detail settings** at the right.

Parallel toolpaths is the default strategy: DeskProto will project a series of parallel toolpaths on the geometry. This is the only strategy that is available in the Free and in the Entry edition of DeskProto.

Detail settings for parallel are the following: **Along X-axis** means toolpaths parallel to the X-axis (so on constant Y), and **Along Y-axis** means toolpaths parallel to the Y-axis. For each of these, two starting points are available: start at the front side versus at the back side (**reversed**), and start left versus start right (reversed).

In addition an **Angle with X-axis** can be entered to create toolpaths that are not parallel to X and Y, but still parallel to one another. The angle value may not be negative: the desired result can be achieved by selecting Along Y-axis with a different A-value. The Angle option is not available for rotation axis machining.



A very powerful detail setting for strategy Parallel is the **Helix** option. This will only be shown for Rotation axis machining (so when in the Part parameters "Use rotation axis" has been checked), replacing option "Angle with X-Axis". it will only be active (enabled) when:

- in the Advance Machine parameters option "A-values may exceed 360" has been checked
- on tab page Movement either Conventional or Climb has been set as Cutting direction
- the parallel toolpaths are either Around A or Around A reversed
- both the Machining area and the Block of material include the complete cylinder (so range from 0.0 to 360.0 for A)

Without this option checked, each toolpath around A will be at one constant X-value. After one complete 360 degree rotation the cutter will move along X to the next X-value and start the next toolpath. So two 90 degree corners for each rotation will result, which will slow down the milling process. The **Helix option** will change that and produce **one long toolpath without sharp corners**: both X and A will show a continuous movement, with the Z-value following the geometry. Perfect for shoe lasts, fishing lures and my more rotary parts.
Using the Helix option combined with Roughing layers (and sorting) will not give optimal results: when skipping what is done in previous layers the Helix path will be interrupted. Best is to use Helix only when finishing.

Second strategy is **Crosswise**. This is the same as creating two operations where one of them uses parallel to X and the other one uses parallel to Y. When you would use two operations there would be redundant calculations though, resulting in longer calculation time: hence this strategy. This option is useful in case the model you want to produce must have a very good surface quality: the staircase effect resulting from the parallel X toolpaths will be removed by the parallel Y toolpaths and vice-versa.
As Detail setting you can choose which of the two directions has to be done **First**, and here as well you can enter an **Angle with X-axis**.

The last detail setting for Crosswise is the **Surface sampling refinement**: see below.

The **Block** strategy combines toolpaths parallel to X and Y to a sort of rectangular 'spiral'. These are probably the most efficient toolpaths, very suited for roughing. So the wizard will as a default select this strategy for its roughing operations.
The Detail settings for block offer two options: **inside out** versus **outside in**, sufficiently explained by the name and the small drawings, and **Angle with X-axis** as just described.
The milling direction can be set on the page Movement tab of this dialog. When machining Outside-in: Conventional will start left to right along X (Counter-Clockwise), Climb will start front to back along Y (Clockwise).
The last detail setting for Block is the **Surface sampling refinement**: see below.

Circular is a completely different strategy as the rectangular base pattern (grid) as applied in the first strategies is not used here. In top view the toolpath shows true circles, projected onto the 3D geometry. For each XY toolpath position the Z-value is calculated using a special radial Z-grid. Of course that strategy can very well be used for round geometries, like rings or cups.
Detail settings are:
Inside out versus **outside in** (same as for block strategy)
Yes or no **machine the corners**: this concerns the area inside the rectangular area to be machined but outside the largest circle that touches all four sides of this area. This option is available only in case the Center has been set inside the operation area: if not it will be "grayed out".
Yes or no make it a **Spiral** toolpath (in top view). This is a great strategy for high speed machines as no sharp angles are present in the toolpath.
Set the **Center** point of the Circle/spiral toolpaths. Standard this point is set in the center of your Operation area, however you can also choose the Center of the Part's material block, or any Custom XY values. These custom values then can be either typed or graphically set using the **Set** button. The center point may even be outside the block.
Here as well the milling direction can be set on the page Movement tab of this dialog: Conventional result in a counter-clockwise direction, Climb clockwise (for outside-in).
If you want you can enter the values for the center in **Translated coordinates**. When you have checked this checkbox the coordinate values are displayed in the coordinates as used on the machine (after Translation has been applied). Checking or unchecking this checkbox does not influence the

toolpaths. It’s just a temporary conversion in this dialog to make setting the centerpoint easier.

Radial is the complement of circular: same Z-grid, however now radial toolpaths (so perpendicular to circular). So the same Detail settings apply here as well, except for the Spiral. And the sequence of the toolpaths now is called **Clockwise** versus **Counterclockwise**.

Offset machining generates toolpaths as offset lines to the border of the area to be machined.
This area can be defined either using option Skip Ambient, using a (freeform) Area to be machined, or using a combination of both. For a complete rectangular area the paths will be similar to the Block strategy, for a round area the paths will be similar to the Circular strategy; the power of the Offset strategy is that this will work for any freeform area to create toolpaths that follow the shape of the geometry.
You can for instance apply the automatically generated freeform area that follows the outer contour of the geometry. The result will be the same as when using Skip Ambient, however now the Border settings can be used.
Typical application example is machining corrective insoles (to be worn inside shoes), which can be done very efficiently with toolpaths parallel to the sole's outer contour.
Detail settings for Offset are:
Inside-Out versus **Outside-In** will be clear, and **Surface sampling refinement** will be explained below.
This strategy produces a number of "parallel" toolpaths, and **Smooth toolpath transitions** concerns the transition to each next toolpath. Normally the last point of the finished path will be connected to the first point of the next path, resulting in two 90 degree angles in the toolpath. On fast machines this will cause the machine to reduce the feedrate at these points. This can be prevented by checking the option Smooth toolpath transitions: DeskProto then will ignore these last points and first points, and instead will connect the last-but-one point of the finished path to the second point of the next path. The result will indeed be a smoother transition. Be careful though: ignoring these two points for each transition may cause DeskProto to remove too much material.
Fill Gaps: the Offset toolpaths follow the outer contour of the area to be machined, each next toolpath at the prescribed distance from the previous one. In the center of the area the paths from opposite sides will meet, and there the distance between the last toolpaths may be larger then the prescribed distance, creating gaps in the toolpath pattern. For flat cutters this will not be a problem, for ballnose cutters at these gaps the cusp (ridge of remaining material) will be higher than for the rest of the geometry. To

prevent this unwanted situation you can use the option Fill Gaps: then DeskProto will add extra toolpaths to also fill these gaps.

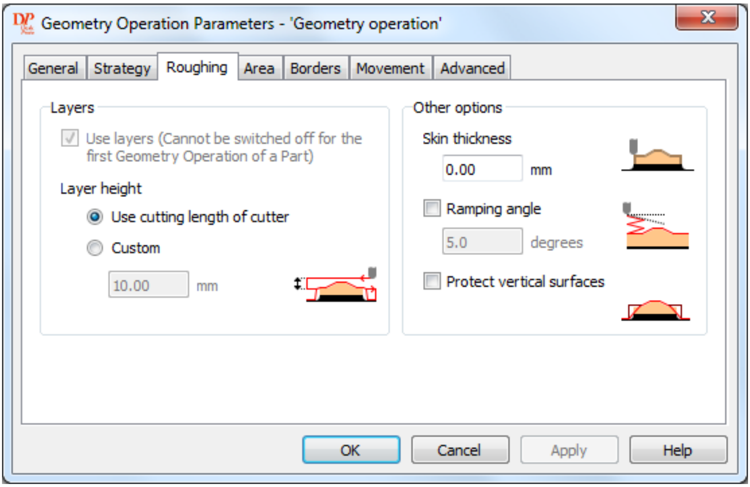
Waterline machining produces toolpaths on a constant Z-level (just like the waterlines over a ship's hull). Such strategy is also called contour machining or Z-plane machining. While the difference with toolpaths on constant X or Y seems small, in reality the difference is huge as completely different calculation algorithms are needed. In the Detail settings an extra parameter is needed: the **Waterline distance** (so the distance between two toolpaths in Z-direction). The XY toolpath distance parameter as set on the General tab page is also used: when horizontal surfaces have to be machined.
Important is to realize that only on the prescribed height levels a toolpath will be present. Imagine for instance a geometry with a horizontal top surface and a hole in that surface (pocket) of 9.5 mm deep. Z=0 is at the top of the part. When you set the Waterline distance at 2 mm, toolpaths inside the pocket will be generated at Z=-2.0 , Z=-4.0 , Z=-6.0 and Z=-8.0 So the depth of the resulting pocket will be 8 mm, not 9.5 as in the Geometry. You can of course calculate a Waterline distance that will give a better result: 9.5/5 = 1.9 mm instead of 2.0 will result in a hole of exactly 9.5 mm deep.
A second waterline parameter is the choice between **Top to bottom**: start at the highest point and work down, and **Bottom to top**: start at lowest Z-level on the outside of the block, and work towards the top.
Finally the Detail parameter **Fill horizontal planes is offered**. This option needs some explanation. As waterline toolpaths have a fixed Z-distance in-between each two toolpaths, at (almost) horizontal surfaces there might be a large distance between two toolpaths. This (horizontal) distance might even be larger than the diameter of the cutter: resulting in islands of material remaining after completing that operation. The option Fill horizontal planes checks where the horizontal distance is too large, and fills the space with toolpaths at a distance as specified on the General Tab. All these in-between toolpaths have the same Z-value, so a visible staircase effect will be the result. You can un-check this option when you use a waterline operation for finishing, after all material has been already removed by previous operations. This will save you much time, both for calculating and for milling.

The last strategy **Contour only** is in fact an additional strategy as it does not machine the complete part: only the outline of the geometry (outer contour) at ambient level is machined. This can be used after some other strategy: to smoothen the model (when DeskProto creates toolpaths that are parallel to the X- or Y-axis, at places where the outside surfaces are almost vertical the contour can show a staircase effect). You can also use it for "Pre-roughing" the material: Give your block the correct outside shape before you start roughing, which will reduce the amount of chips.

You can use detail setting **Offset** to have the Contour line toolpath keep a certain distance to the geometry: add a skin, however then only for X and Y. Just as for a Roughing skin defining an offset will change the pre-set Precision values, as the offset is achieved by using a different cutter diameter for the calculations.
Detail setting **Ignore enclosed contours** does exactly what its name suggests: when a hole is present in the geometry the standard result will be a contour inside the part's outer contour (nested contours). Checking this box will make DeskProto create a toolpath only for the outer contour.

The **Surface sampling refinement** parameter that is offered for some strategies is meant for advanced users, as normally the default values are appropriate. After checking the box you can use the **Settings** button. As a result the Surface Sampling refinement dialog will be shown that allows you to fine-tune the cell-size of the Z-grid, which will influence the calculation precision.

Roughing parameters



Roughing stands for quickly getting rid of most of the material without milling very precisely. So after an roughing operation you will need a second operation which machines the same area more accurately: the finishing operation. In DeskProto you can use an operation either for roughing or for finishing, so if you need both you will have to **Add** an operation in the General Part parameters.

When roughing you want to make sure that the cutter does not cut too deep into the material: deeper than the cutting length of the cutter is not permitted, though for hard materials that will still be way too deep.
The roughing option **Layer Height** maximizes the cutting depth: instead of trying to remove all material at once, this will be done layer by layer. The default layer height equals the whole Cutting length of the cutter. In most cases it is preferable to use a smaller **Custom** layer height, as with a tough material you do not want the cutter to use its total cutting length. As a rule of thumb: set the Layer height equal to the cutter's (flute) thickness.

The first Geometry operation of each part always uses layers. This is done automatically and cannot be overruled: DeskProto does not allow the cutter to remove more material than possible for the cutting length of the cutter. For subsequent operations you can un-check **Use Layers**: the cutter will then machine at full depth all the time. In case your are sure that sufficient material has already been removed to allow skipping the layers (for instance by a Bitmap Operation) you can add an extra Geometry Operation to the part, before the current operation, and make it invisible (an unused "dummy" operation).
Note:
The first layer starts at the top of the material block. When your block is higher than the part you can use a custom Material block having a higher Maximum Z value.

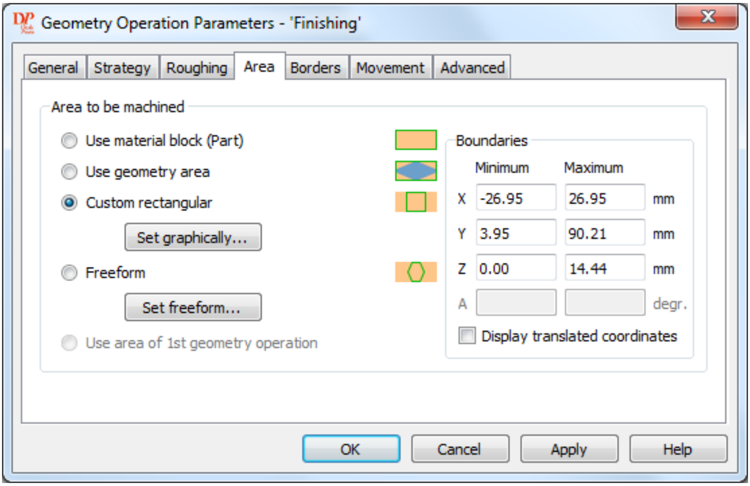
Entering a **Skin Thickness** results in a model which is thicker: a skin is added everywhere around the model, as an extra allowance. In this way the chance that the cutter takes away too much material is reduced (this can happen as roughing typically will be done using a low precision, and as the cutter will vibrate and may bend during roughing). Using a skin also improves the resulting surface quality, as then during finishing the tool will remove the same (small) amount of material all the time. Internally DeskProto processes the skin by applying a different size (and shape) cutter.
It is possible to set the Skin on a negative value, making the resulting part too small. This is interesting in some special cases, like for creating electrodes for EDM machining (spark erosion), or for machining a foam core to apply modeling paste on for the final cut to size.
Of course entering a value of 0.0 means that no skin will be applied.
Warning on skin use:
The skin is also applied on vertical surfaces, which in case of high vertical walls may lead to a problem during finishing. When finishing the cutter

machines on full depth, so it has to take off the skin of the complete wall in one go. In case the wall is higher than the cutting length of the tool this is a problem for which no automatic solution is available yet. An easy workaround to solve it is to add an operation using the waterline strategy, milling from top to bottom, and machine these waterline toolpaths first.

The **Ramping angle** is used when starting to machine. The cutter will then first move to the correct XY position: exactly above the first point to be machined, on the Free movement height Z-level. Then the cutter will move down to that first point to be machined, which is normally done in one vertical downward movement.
Such vertical downward movement is not ideal: many cutters do not like that, and the chips cannot escape the deep round hole that is created. Entering a Ramping angle makes DeskProto replace this vertical movement by a series of ramping movements: go down along a sloping line. See the small picture in the dialog for this option. You can set how steep this sloping line needs to be by entering an angle value in degrees: this is the angle between the sloping line and a horizontal line.
- Ramping is applied when moving down to the first point of an Operation (in case of multiple layers: for every layer). It is not applied on other downward vertical movements.
- Ramping uses only the space that is available above the first horizontal toolpath. In case that first toolpath is very short then ramping is not possible.

The option to **Protect Vertical Surfaces** is useful when roughing with a large value for the Stepsize along toolpath. In case the steps are large they may not "see" all geometry in-between and remove too much material. This may happen in case of vertical or steep surfaces in the geometry. Checking this option will replace any tool-movement steeper than 45 degrees by separate horizontal and vertical component movements. Take care: *on many machines this may cause unwanted vibrations as smooth paths are replace by staircases*, so only check this option in case needed.
This resembles the algorithm as used in Vertical surfaces in the Advanced Operation parameters with a height/step ratio of 1, however this latter advanced option is more intelligent.

Area parameters



This tab page offers almost the same options as the Material tab page of the Part parameters.
Note that the functionality that is offered is different though ! The Material block defines the size of the complete Part: any geometry outside the boundaries of the Part will be removed. The Operation area only defines the **area to be machined**. So it limits the toolpaths for this Operation, to be used for instance when there is a small area of the part that is very detailed and needs to be machined with a smaller tool in an extra operation. Any geometry outside this area (and within the part) will not be damaged.
As you can see in the icon pictures, the bounding box of the area is be drawn in **green lines.**
The area definition also applies to the Z-axis, allowing you to set a maximum machining depth for the area.

The default option here is **Use material block (Part)**. This means that the cutter will machine the whole part you want to create.

When in your project you have combined various types of CAD data (geometry, vector and/or bitmap) it may be handy to use the bounding box of the relevant data type as area:
In a *Geometry Operation* this can be done by choosing option **Use geometry area**.
In a *Bitmap Operation* this can be done by choosing option **Use bitmap area**.

Two types of custom area are available:

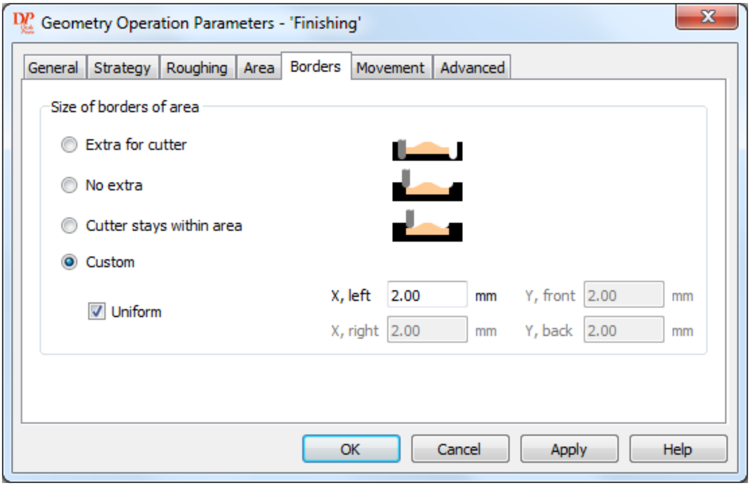
Custom rectangular sets a rectangular block as area, just as for the material block. This block can be defined by entering the min and max **Boundaries** in the edit boxes at the right, or graphically.
Checking the option **Display translated coordinates** changes the numbers that are shown for the area boundaries: these will now be in workpiece coordinates as used on the machine, so after Translation. Only the numbers shown here for input are changed, not the actual coordinate values in the toolpath: it is just a temporary conversion on screen for setting the boundaries more easily.
The button **Set graphically** pops up the Set Area dialog that makes it very easy to set any area using the mouse.

The second type is the **Freeform** area. This option allows the use of areas that are not rectangular, for instance a circle or any freeform contour (freeformed in top view). Button **Set freeform** pops up the Set freeform area dialog.
For a freeform area the Min and Max X and Y values shown as Boundaries relate to its bounding box. The min and max Z can be set as for a rectangular area.

Final option is **Use area of 1st geometry operation**, of course active only for second and further geometry operations in the part. This is a handy option for instance when a complex Freeform area needs to be used both for Roughing and for Finishing.

The area may not be larger than the block of material that has been defined for the part, for any of the axes.

Borders parameters



As has been explained already, in DeskProto the toolpaths normally cover a rectangular area. In most cases this area needs to be a bit larger than the minimum and maximum values of the part, allowing the tool to move all around the geometry: to machine the outside surfaces. This extra area at the 4 sides of the rectangle is called the **Border area**, and in this Tab you can influence the size of the border area. The Z-level used for the border area is set at the Ambient tab of the Part Parameters.
Note that you can also make the material block or the area larger to add extra area to be machined.

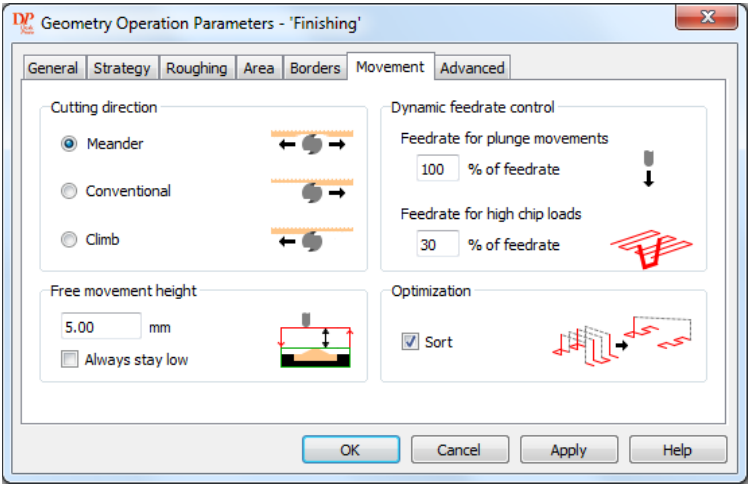
The default option is **Extra for cutter**, which sets the border area size exactly to what is needed to let the cutter go around the model, in order to machine all outside surfaces of the part.
In the four edit boxes you can see that this value matches the Radius of your cutter. The cutter will of course cut at its full diameter, however here we consider the position of the *center of the cutter*. And for this center the max distance to the geometry is the *Radius* of the cutter.
The size of the Extra for cutter area is different when you have set an Angle in the Strategy sub-settings (toolpaths at an angle with the X-axis or Y-axis). We have found that the border area then needs to be larger, as otherwise the cutter cannot completely move down on all sides of the part. How much larger depends on the Angle that has been set.

The second option, **No extra**, keeps the cutter positions (for the centerpoint of the cutter) exactly inside the [area to be machined](). Using this option it is still possible that the tool cuts away material that lies just outside the area, as half of the cutter reaches outside. DeskProto will anyway check that any geometry that belongs to the Part is not damaged.

The option **Cutter stays within area** means that the whole cutter will stay inside the area to be machined. This option is used by the Two Sided Milling Wizard, to make sure that the area that is machined is exactly the same for all operations, no matter which cutter is used.

The last option, **Custom**, makes it possible to define the border sizes yourself, using the four edit boxes. The values may be positive values as well as negative. For negative values there is a limit though, as obviously some area that can be machined has to remain.
The use of the checkbox **Uniform** will be clear: it forces all four edit boxes to have identical values.

Movement parameters



The **Cutting direction** is important for the surface quality of the prototype. The default direction is **Meander**, which means that (for parallel toolpaths) the first movement is from left to right, the second from right to left, etc. The tool keeps cutting all the time, so meander is the fastest option.

However, on the surface of the model you may see a difference between the movements going from left to right (L-R) and the movements in the opposite direction. The surface will be smoother when all movements go in the same direction. Obviously there are two possibilities here: L-R and R-L. The words **Climb** and **Conventional** refer to the relation between the direction of milling and the rotation direction of the tool (normally clockwise): see the small drawings in the dialog.
Note that choosing the option Climb or Conventional in fact does not guarantee it to be used all the time: when machining a downward sloping surface it is possible that in fact the back of the tool cuts, thus reversing the actual cutting direction.

Meander is not possible for all strategies: for strategies Block, Spiral, Offset, Waterline and Contour meandering cannot be selected: for these strategies it would not make sense. That is, unless the option Sort is active. When sorting, also for these strategies some areas may be present where the toolpaths can be optimized by making them meander. In these cases DeskProto offers the Meander direction with the remark "only optimizes sorting".
You will have to test though whether or not this works for your project: in some special cases selecting Meander will lead to extra positioning movements, so to a longer toolpath.

The **Free movement height** is the Z-level at which all ‘non-cutting’ tool movements will be executed. This is for rapid positioning movements over the part, for instance from the home position to a position above the first point to be milled. The Z level you enter here is the number of units (mm or inches) above the top of the material block. Only positive value are allowed: in case of a negative free movement height the model and/or the cutter might be damaged.
When a Skin has been set (Roughing) the free movement level will be the Skin-thickness higher

This Free movement level (ZFree) value will be used in three ways:

- The first point and the last point of the toolpath for each operation are located ZFree mm/inch above the top of the Material Block. This is needed to make sure that the cutter is high enough when moving to and from these points, so instance to a next operation. These movements are done in Rapid mode, which is permitted only above the block.
- The positioning moves during an operation are performed at ZFree mm/inch above the max Z of the Operation Area, or (in case that is lower) above the max Z of the geometry within the XY limits of the Operation

area. This is done in order to speed up the process when machining some detail area at a low Z level.

- When the option **Always stay low** is checked, the positioning moves are performed at ZFree mm/inch above the highest point of the geometry and/or bitmap relief over which the cutter moves. For machines with a slow Z-axis this will save a lot of time. Also when on your machine switching between Rapid and Normal movement is slow this will save time, as these movements below the top of the block are done at normal Feedrate. Always stay low is not possible when Roughing layers are used.

Not all positioning movements are done on this Free movement level, as for small distances this is not needed. The following movement types are applied -- use type 1, if not possible use 2, if not possible use 3:

1. Direct movement.
The cutter moves in a straight line between startpoint and endpoint.
This happens when:
- **the distance to be traveled is less than 1.4143 times the Toolpath distance**.
As toolpath distance is used: for Geometry and Bitmap the Precision value, for Vector the Pocketing Stepover distance (or its default, being 50% of the diameter of the cutter tip).
or:
- (only for 2D Vector operations and for strategies Contour and Waterline, where the toolpath is at constant Z) **the distance to be traveled is less than the diameter of the cutter tip**.
In both cases as tip value is used: for conic cutters the tip diameter, for other cutters the flute diameter, in case no flute is present the shaft diameter. So for conic cutters with a sharp tip this diameter is 0.0 and thus for such cutter no direct movements are allowed here.
In both cases DeskProto will also check if no intermediate Z-grid positions with a higher Z-value are present, as then direct movement is not possible.

2. Lowered Z-height.
The cutter rises a bit, moves in a horizontal line and then goes down to the endpoint, at normal Feedrate.
The Z-height is Z-Free mm/inch above the highest point of the geometry below this line (this does not work for a bitmap relief).
This happens when:
- **the distance to be traveled is less than 1.4143 times the Toolpath distance** (so when in type 1 an intermediate position with too high a Z-value was detected).
or:

- **the distance to be traveled is less than 10 times the smallest of Toolpath distance, Cutter tip diameter**
or:
- **the distance to be traveled is less than the cutter-shaft-diameter**
or:
- **in the Operation parameters the option "Always stay low" has been checked**.
Exception: "Always stay low" is not (yet) possible when roughing layers are applied, as then for some strategies the lowered positioning movement may travel through un-machined material.
For Vector operations without geometry checking Always stay low will only change the Feedrate, not the Z-height.

3. Free movement height.
The cutter rises to Free movement height for a positioning move at Rapid speed.
The Free movement height is Z-free mm/inch above the top of the operation area or, if that is higher, Z-free mm/inch above the highest point of the part geometry that the cutter travel over. The first point and the last point of a toolpath are at Z-Free mm/inch above the top of the Material block.
This happens in all other situations.

One last remark about the Free Movement height, for advanced users. While in this dialog the user-interface offers the option Always stay low, in addition a **hidden option** is present, called **Never stay low**. This option cannot be checked in the user-interface, in order to use it you need to add a new String value to registry key
HKCU\Software\Delft Spline Systems\DeskProto\7.0\Settings
This new string value needs to be called *NeverStayLow* and needs to have the string "true" as content.
When this registry key value being "true" is found, DeskProto will let the cutter rise to Free movement height for every positioning movement. So also when proceeding to the next toolpath. This hidden option is used for a few applications where roughing is done by a saw (sawblade, chain-saw), as that cannot move sideways.
When the string is "false" (or in fact any other text) nothing will happen.

The **Dynamic Feedrate** control is an advanced option of DeskProto: even many so-called high-end CAM software packages do not offer this type of functionality. It means that DeskProto is able to reduce the feedrate when needed, thus making it possible to select a high overall feedrate without the danger of breaking your tool at the critical points that will be encountered.

Two separate options are offered. In both cases you can enter a percentage for the maximum feedrate reduction applied; and in both cases DeskProto will choose in-between feedrates whenever possible, thus always running at optimum feedrate. Note that both options can be combined, in which case for certain movements both reductions will apply, resulting in a very low feedrate.

With the **Feedrate for plunge-movements** it is possible to decrease the feedrate when the cutter moves downwards. This may be needed when machining in metals, as fast plunge movements may damage the cutter (many cutters have problems with drilling). It is expressed as a percentage of the normal feedrate for this operation.
The rate you enter here will be used for movements that go down along an angle that is more than (steeper than) 30 degrees.
For downward movements along an angle less than 30 degrees the reduction will be smaller: DeskProto will apply the rate as specified + 20. So when you have set the plunge rate to 40 %, these movements will be reduced to 60 % of the normal feedrate.

It's also possible to decrease the **Feedrate for high chip loads**, which happens when the cutter has to remove much material. Due to DeskProto's parallel toolpaths approach, normally the cutter only has to remove a small slice of material: a thickness of only the distance between two toolpaths (the stepover). However, in certain cases the cutter has to machine away material over the full flute diameter, which is a much higher chip load. For instance for the first toolpath (as often the block will be a bit larger than needed); also when for the first time entering a pocket in the model (so when the tool suddenly has to machine much lower than during the previous toolpath). The chip load is even higher in such cases as the chips cannot easily spread out but will be stuck in the groove that is machined.

In these cases the feedrate will be reduced, the actual reduction depending on how much deeper the tool has to cut compared to the previous toolpath at that position. Reduction is applied according to the following rule, where D is the cutter's flute diameter and Rate is the percentage that was entered. The column "Example" shows the resulting actual reductions in case a value of 20 % resp 70 % has been entered.

Difference in depth:	*Reduction percentage:*	*Example:*
For rates below 60 %:		
Until 0.1*D	100 %	100 %
0.1*D - D	Rate + ½ * (Rate)	30 %

D - 2D	Rate + ¼ * (Rate)	25 %
more than 2D	Rate	20 %

For rates of 60 % or higher:

Until 0.1*D	100 %	100 %
0.1*D - D	Rate + 2/3 * (100-Rate)	90 %
D - 2D	Rate + 1/3 * (100-Rate)	80 %
more than 2D	Rate	70 %

So (explanation of the table above): reduction is applied only in case the cutter machines **deeper** than in the previous toolpath, with a margin of 0.1*D (10 % of the cutter diameter D). When between 0.1*D and D deeper the first reduction step is applied, between D and 2D the second, and when the difference is more than 2D the full reduction percentage is applied. In the toolpath drawing on screen, the paths at reduced feedrate will be drawn in a slightly different color (purple instead of red).

The option Reduced feedrate for high chiploads is not available for all strategies.
And these Dynamic Feedrate options are of course only possible on machines that can set the Feedrate from the PC, which is determined by the Postprocessor.

The Number of Reduced Initial Paths.
As mentioned above this option will also reduce the feedrate for the first toolpath of an operation. When machining a flexible material (for instance foam) it may be needed to reduce the feedrate for more than just this one first toolpath. This has been implemented a "hidden feature", using a registry entry called *NumReducedInitialPaths*, which sets the number of initial toolpaths to be reduced. The registry entry will work only for strategies Parallel, Crosswise and Block.
It can be switched on by adding a new value to registry key HKCU\Software\Delft Spline Systems\DeskProto\7.0\Settings\
Add dword (32-bit) value NumReducedInitialPaths and give it a value from 1 (default) to 9.
Only edit the registry when you are qualified and know exactly what you are doing !

DeskProto is able to **Optimize** the toolpaths using a **Sorting** algorithm. It will then ignore the sequence as defined by the Strategy, and instead try to complete series of tool-movements that are located close to one another. For instance when machining the Picture frame sample geometry using strategy

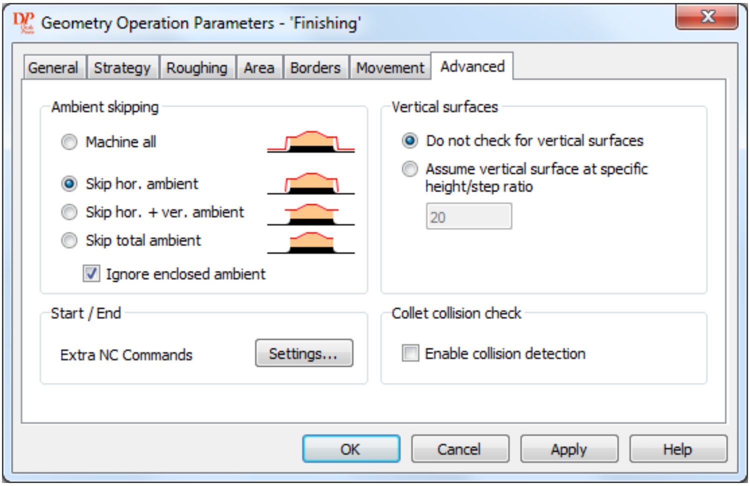
'Parallel to X' and skipping the empty inside of the frame, the cutter would normally need to make positioning movements from the left to the right side of the frame all the time. Sorting will make DeskProto first complete one side, and only then start with the other side: preventing many positioning movements. Sorting can especially save much time in case many roughing layers are applied.

Background information about the sorting algorithm that is used, written by the programmer:
In DeskProto each toolpath is a series of small lines (linear interpolations, or G1 movements), where each movement is a straight line between two points. A series of connected movements (in fact a polyline) is called a chain. A chain ends when a positioning movement (go up, position, go down) is needed.
The third concept that is used is the tour: a tour is one complete path for the chosen strategy, so when not interrupted for some reason. For strategy parallel a tour is one complete path from left to right, for block one complete path around all four sides, etc. Subsequent tours are more or less parallel to one another.
Sorting compares the start point and endpoint of each chain with all chains on the next tour. If on that next tour a adjacent chain (so both start points and end points close) can be found that is closer that the next chain on the same tour, then DeskProto will jump to the chain on the next tour.

For the waterline strategy the situation is more complex. Here DeskProto uses the bounding box of each chain. If the bounding box of a chain does not overlap any other bounding box (on the same Z-level) then it is an isolated chain. After machining such isolated chain DeskProto will jump to an isolated chain on the next Z-level (and on the same location). So when machining a city with church towers, sorting will make DeskProto complete the towers one by one, instead of finishing a complete Z-level before starting on the next Z-level.

For the Contour-only strategy sorting can change the sequence of the contours - of course only in case more than two separate contours are present.

Advanced parameters



Ambient skipping concerns the ambient area. Basically DeskProto always machines a complete rectangular area, due to its parallel toolpaths approach. In case the model only takes up a small part of the area, this might lead to unneeded extra chips and unneeded extra machining time. Also: when all material around the model already has been removed in a previous roughing operation there is no need to again machine the ambient area when finishing.
In such situations you can optimize the toolpath by using the Ambient skipping option.

- By default this option is set to **Machine all**: the complete area will be machined.
- **Skip hor. ambient** means that all extra horizontal movements on ambient level, from the model to the border and back, will be skipped. The tool will still go down to ambient level around the model. You can for instance use this for finishing, when all ambient material has already been removed by the roughing operation.
- **Skip Hor. + Ver. ambient** means that both horizontal movements on ambient level and the vertical movements towards ambient level will be skipped. For ballnose cutters then still a groove round the part will be visible: when only a small part of the cutter is above the model the tip of the ball will have a lower Z-value than the part geometry at that point.
- **Skip Total ambient** means that the center of the cutter will remain above the geometry: all positions with the center of the cutter above ambient area will be skipped.

The icon pictures on this tab page illustrate the differences between these four skipping options.

An extra option that can be checked here is called **Ignore enclosed ambient**. When checked, only the ambient around (so on the outside of) the part will be skipped. The enclosed ambient area, like for instance within holes in the model, then will be machined.

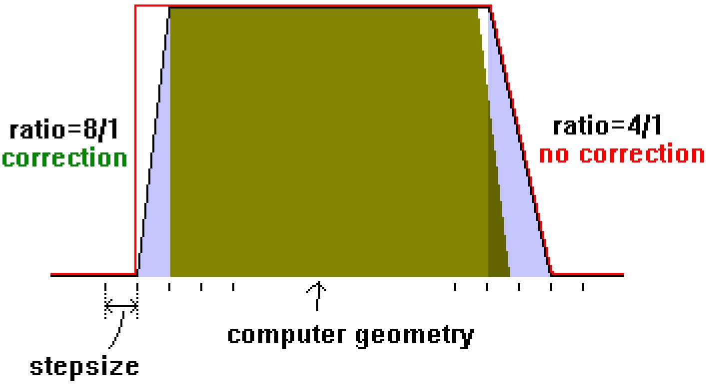
The algorithm used by DeskProto to calculate toolpaths does not really support true **Vertical surfaces**. DeskProto calculates its toolpaths based on a Z-Grid, connecting points of the grid to form the toolpath. For each XY position in the grid only one Z value is available, which means that vertical movements are no possible: that would require two points with the same XY and a different Z. Each tool-movement will thus contain both a horizontal and a vertical component, the horizontal component being the stepsize along the toolpath. As result a vertical surface in the geometry may be machined having a small angle in the resulting part.
BTW this is not true for all strategies: the above does not apply when using waterline toolpaths

In case you do need a model with true vertical surfaces (and do not want to use waterlines) you can achieve that by using and fine-tuning this option. You can let DeskProto assume a vertical surface in case the 'toolpath-line' is steeper than a certain angle. The angle is defined by the Ratio between the height and horizontal distance (stepsize) of one movement in the toolpath, and in the edit box you can set the **Height/step ratio** to be used. See the illustration and examples below.

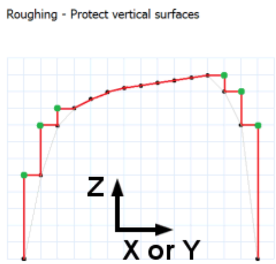
For every movement DeskProto will check this ratio, and for a movement that exceeds the ratio (so when the movement is steeper), DeskProto will insert an intermediate movement: the movement will be split into a horizontal and a vertical component, to be executed sequentially. The result will be a vertical surface in the model, which otherwise would have been angled.

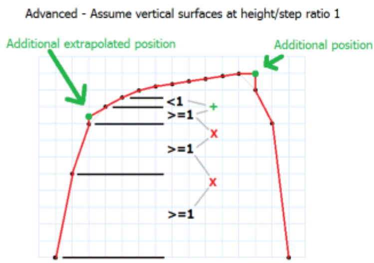
Example: **Vertical surface ratio = 8**
The image below shows a geometry with a vertical wall on the left and a sloped wall on the right. The Stepsize has been set to a value that is 1/8th of the total height of the geometry. You can see 2 toolpaths: one drawn in black, one in red. The black one is the one that is first calculated. When you then check the vertical surface ratio option and set it to 8, the toolpath will be changed to the red one. So on the left side the toolpath will have been corrected whereas on the right it will not, because on the left the height/step

ratio is 8/1 whereas on the right the height/step ratio is 8/2 = 4/1. In the drawing these two ratio's are shown as purple triangles. On the left DeskProto assumes a vertical surface and changes the toolpath, on the right side not. For a vertical surface DeskProto will add one cutter movement, splitting the original angled movement into a vertical and a horizontal part. These two movements will be output in a sequence that leaves extra material on the part.



The implementation of the Vertical Surface setting is different for tab page Roughing (image on the left) and tab page Advanced (image on the right):





In the **Advanced** implementation the extra position will be added only for the *first* movement (or the last for an upward toolpath), meaning that this will only happen at the transition point between a "horizontally oriented" section of the path and a "vertically oriented" section. See the illustration above right, showing the result for a height/step ratio of 1.
In the **Roughing** implementation a much simpler version of this algorithm is applied: with a fixed ratio of 1, and for all movements (so for all lines of 45

degrees or steeper): the illustration above, left. The result will be a staircase shaped toolpath and a lower surface quality, so select this only when needed.

For parts without vertical surfaces this functionality is not needed, so for such models choose **Do not check for vertical surfaces**.

It may be needed to experiment a bit with this option in order to find the best ratio for your geometry. Take care when creating parts designed with a draft angle: in case this option is used the result might be that all draft surfaces will be made vertical **!**

Example 2: now with numbers.
Assume that your part contains a vertical surface of 15 mm high, and that you have set the Stepsize to be 1 mm. Now a toolpath over this surface has been generated that includes a horizontal component of 1 mm (this stepsize), however that you want to be really vertical. The default height/step ratio of 20 then means that when this tool movement goes up or down over more than 20 mm it will be split up into two separate movements (horizontal and vertical). As your desired vertical wall is only 15 mm high you need to set the ratio to (say) 14 to make DeskProto change these toolpaths.
This will work for any Stepsize: when you have for instance selected 0.0262 inch, then the height/step ratio of 20 means that when the tool movement will be split when it goes up or down over more than 20 x 0.0262 = 0.524 inch.

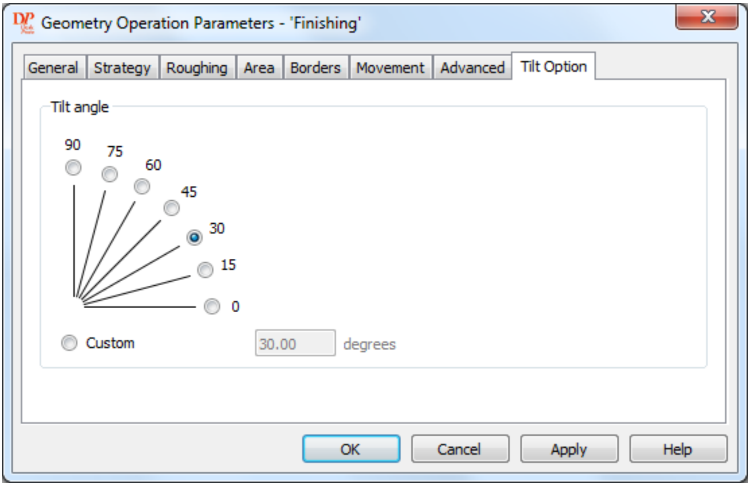
Note that this vertical surfaces check does not work for conical cutters as the steepness of the toolpath is analyzed (for conical cutters the toolpath never can be steeper than the Angle of the cutter).

Start / End offers the option to add extra commands to the NC program before the operation toolpath starts and/or after it has ended. These commands can be movement commands and/or user defined commands. They can be used for instance to make the cutter move to a safe position before rotating the 4th axis. Button **Settings...** will open the Operation Start/End settings dialog.

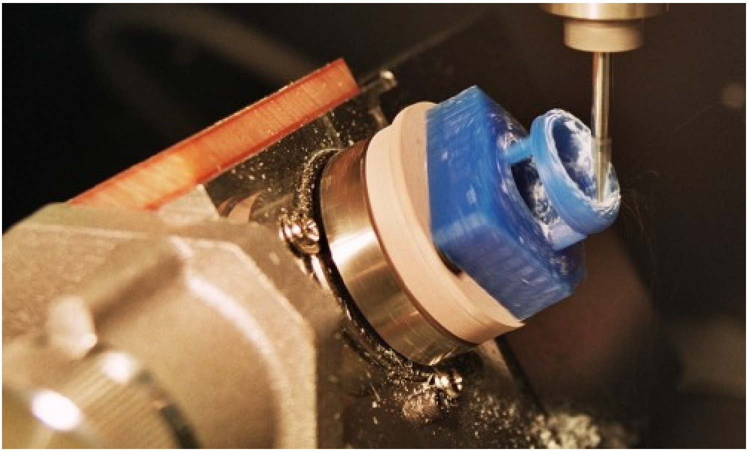
The **Collet collision check** is a very useful option for high models, especially with high vertical or steep surfaces. The problem with such models is that in case the vertical wall is higher than the free length of the tool, the milling machine's collet will damage the model (DeskProto only compensates for the geometry of the cutter, not for the machine). Checking this option means that DeskProto will let the tool move away from the model in such

case, preventing the collet to collide with the model. Obviously the resulting model will no longer be correct, as the material that cannot be reached by the cutter will be left on the bottom of the vertical wall. However this is much better than letting the collet damage the top of the model, as the excess material can later be removed with a different cutter or by hand.
The diameter of the collet can be defined at the Machine parameters dialog. Note that when using this option the border area will be enlarged, to enable the complete collet to move down on all sides of the model.

Tilt option parameters



The **Tilt Option** tab is visible **only** in case in the Part parameters the option "Use rotation axis tilt option" has been checked. It is meant for a special type of 5th axis: the manually controlled B-axis as present on some small wax milling machines for jewelers. Like on the Roland JWX-10 "Jewela" machine, as shown in the illustration below:



In the bottom-left corner of this photo you can (just) see a rotation knob: after unlocking the rotation mechanism you can use this knob to manually tilt the complete rotation axis (A-axis) unit. This machine supports locking on fixed intervals of 15 degrees.
As this manual rotation is round an axis parallel to Y, this rotation is called a B-axis. The total number of axes then is 5 (X, Y, Z, A and B), so this is in fact a (very primitive) 5-axis machine.

The **Tilt Angle** that you can enter here is the rotation value that will be used for this Operation. As you can see in the illustration, this rotation offers you the possibility to machine in places where the cutter normally cannot reach: for instance the inside of a ring model.
In addition to these 15 degree intervals you can also enter a **Custom** rotation value here.

Note that the position of the part after this rotation depends on the distance between the part and the actual B-axis (so the axis of rotation). The larger this distance, the more Z-movement while rotating. This distance can be set on the Zero-point tab of the Part parameters, as at angle 0 this distance is in fact the X-translation. You will see that this Zero-point tab looks differently when this Tilt option has been selected.
Setting the correct X-translation here is important: only with a correct translation all resulting toolpaths from operations with different tilt angles will be correctly positioned (all with the same Workpiece zero point).

The Tilt option is available only for Geometry operations, so not for Vector and Bitmap.

4.3.9 Bitmap Operation Parameters

DeskProto features three different types of Operations: this dialog is for the Bitmap Operation -- in addition also dialogs for a Vector Operation and for a Geometry Operation are available.

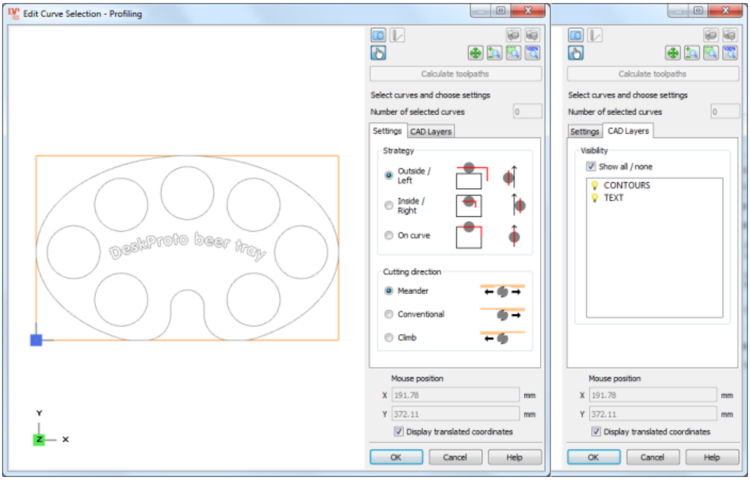
The Bitmap Operation parameters are identical to the Geometry Operation parameters (except for the Tilt option: that is not available for bitmap). The Bitmap Operation applies them on the Bitmap relief, the Geometry Operation applies them on the Geometry.

So **for information on any of the Bitmap tab pages, jump to these Geometry operation Help pages:**
General
Strategy
Roughing
Area
Borders
Movement
Advanced
Tilt Option (not available for bitmap operations)

This dialog can be reached via the Parameters menu, third option.
Or you can double-click on an operation-item in the project tree (one of the third level items).
Or right-click on an operation-item and select Operation Parameters in the context-menu.

This same dialog is used for the Default Bitmap Operation parameters, only with an extra button **Restore DeskProto defaults** to reset the original default parameters.

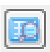
4.3.10 Select Curves



Purpose of this dialog is to select the vector curves to be used for this operation, or to edit that selection.
It is present in three different versions: once for **Profiling**, once for **Pocketing** and once for **Drilling**. All three versions offer the same functionality, except for the options in the "Settings" field. The above image shows the version for Profiling, the settings for the other versions are shown below.

The Select Curves dialog can be reached by pressing button "Select..." on respectively the Profiling tab, the Pocketing tab and the Drilling tab of the Vector Operation parameters.

A top view drawing is present (no other views), in which all loaded curves will be drawn in **gray**.



Button to open dialog Items visible for Set Graphically, in which you can select which items should and should not be included in this drawing. It will be clear that item "Vector curves" now cannot be switched off.

Button Toolpaths will show (and if needed first calculate) the toolpaths in this top-view drawing. Pressing the button again will again remove the toolpaths from the drawing. This is of course possible only after one or more curves have been selected.

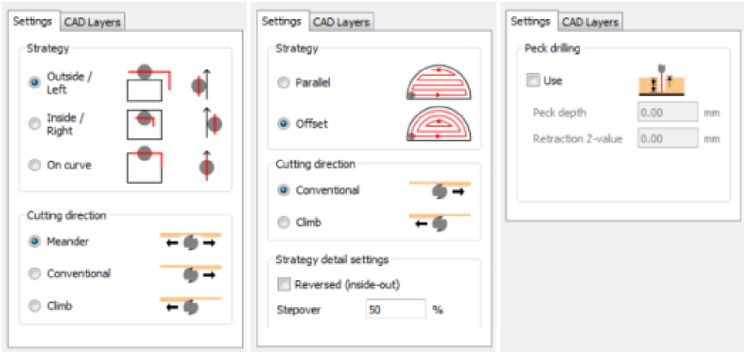
The use of buttons Previous view and Next view will be known from the Toolbar.

The leftmost button on the second row is the **Select** button. It is one of the five buttons to set the current Mouse function. Four of these are known from the Toolbar, this leftmost button is the **most important button in this dialog**. When it is active you can select a curve by pointing the cursor at it using the mouse: when pointing correctly the curve will turn **purple, for "selectable"**. You then can select it by clicking: its color will then change from gray to **black, for "selected"**.
To add more curves to the selecting you need to keep the Shift key (on the keyboard) pressed (if not a next click will undo the previous selection). Undoing the selection can be done by clicking on the white background (unless the Shift key is pressed).
On the third line DeskProto will show how many curves have been selected.

Not all curves can be selected:
- Profiling will not accept single points
- Pocketing will accept only closed curves
- Drilling will accept only single points, +-signs sized equal to the cutter diameter and circles that equal the cutter diameter.

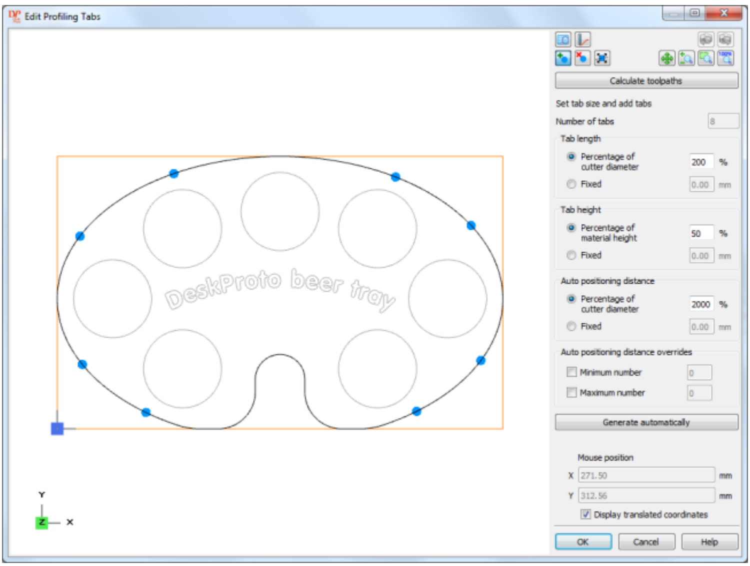
Curves that are not acceptable will be drawn in **light gray, for "can not be selected"**.

A very handy tool is available under tab **CAD Layers**. Here all layers in the available Vector data are listed (if any), with a light bulb icon that can be used so show or hide the curves in that layer.
Many DXF files contain layers, meant to make it easier to access the data by viewing at it layer by layer. Like in this DeskProto sample file *2D_DpBeerTray.dxf* : one layer contains all contours, the second layer contains all text. Viewing the curves in this dialog per layer makes it very easy to select the correct curves, as curves that need the same settings typically will have been grouped in one layer.
Note that these CAD layers are completely unrelated to the Roughing layers that DeskProto uses when roughing.

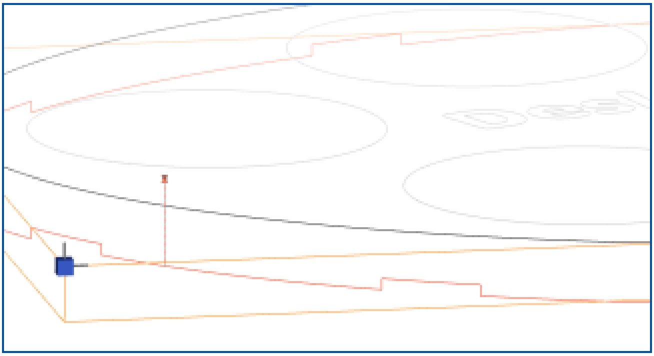


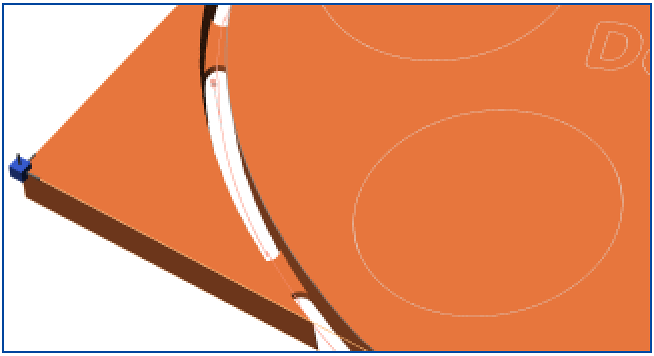
Under tab **Settings** you can set the most important parameters for that toolpath type: the above image from left to right shows the settings for Profiling, Pocketing and Drilling. These are the same as present in the Vector Operation parameters, making them also available in this dialog will allow you to quickly see the resulting toolpaths for any setting.

4.3.11 Set Profiling tabs



When machining a 2D vector profile in sheet material you can set the depth of the toolpath equal to the material thickness, making a cut that goes completely through the sheet. When cutting a closed profile this will result in a separated part: the material inside the profile no longer is connected to the rest of the sheet.
When the material is clamped only at it's corners, that separate part will be completely loose, may go wild, and can be damaged and/or cause damage. Which is of course an unwanted situation.





DeskProto offers **Profiling tabs** to solve this problem. Such tab keeps both sides of the toolpath connected by raising the cutter for a small section of the toolpath, leaving a small bridge of material that later can be manually removed. The two images above clearly illustrate how this works: left the toolpath, right a simulation.

This tab page is about Vector tabs, made by changing the machining depth in a 2D toolpath. Not to be confused with Geometry support tabs, made by adding small blocks to the geometry.

The dialog **Set Profiling Tabs** allows you to set the number of tabs, the location of each tab and the tab size. You can open this dialog on the Profiling page of the Vector Operation Parameters. The drawing will show a top view of all vector curves, with the selected curves drawn in black. Profiling tabs can be set only on curves that have been selected for profiling, in the same operation. The location of a tab is shown by a blue dot, see the screenshot above.

The buttons in the top right corner offer the following options:

Button to open dialog Items visible for Set Graphically, in which you can select which items should and should not be included in this drawing. It will be clear that item "Vector curves" now cannot be switched off.

Button Toolpaths will show (and if needed first calculate) the toolpaths in this top-view drawing. Pressing the button again will again remove the toolpaths from the drawing.

Button Add tab will be active as default when you open this dialog. It sets the mouse function to adding tabs: move the cursor over the drawing to highlight one of the black profile curves (it will become violet) and left-click: on that location a blue dot will be added on the profile.

Button Delete tab sets the mouse function to deleting tabs: move the cursor over the drawing to highlight one of the existing blue circles, then left-click in order to remove it.

Button Move tab sets the mouse function to Moving tabs: move the cursor over the drawing to highlight one of the existing blue circles, then left-click: while keeping the mouse button pressed you can move the blue dot. Moving is possible only along the profile curve.
The meaning of all other buttons is the same as on the main screen, so needs not explanation here.

Other options on the right side of this dialog:

Tab length
The length of the tab is the distance along the curve along which some material will be left. That is not the same as the length of the elevated toolpath fragment: that needs to be increased by the cutter radius on both ends.

You can set the tab length either as a **Percentage** of the cutter diameter (so it will work for any size cutter and any size part), or as a **Fixed** distance in mm or inch.

Tab height
The height of the tab is the thickness of the remaining material. Easiest is to define this thickness as a **Percentage** of the sheet thickness (so of the material block height), second option to define it as a **Fixed** value. A thickness of 0.0 (or 0%) is not permitted, and the tab may not be higher than the material block (so max percentage is 100%).

The above options have shown how to manually set support tabs. It is also possible to make DeskProto set tabs automatically, using the following parameters:

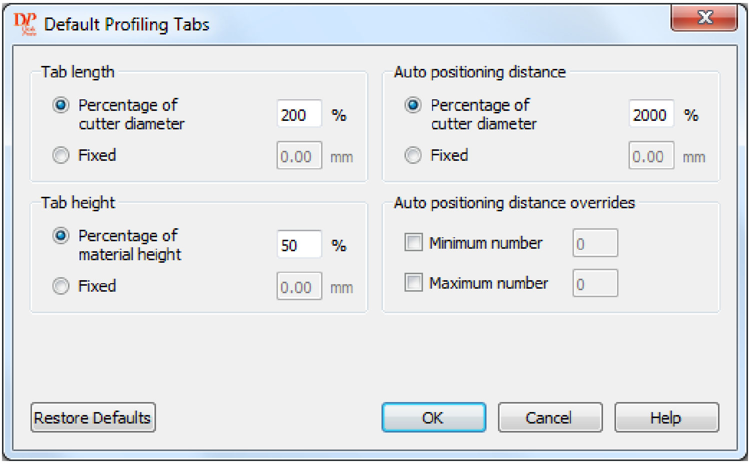
Auto positioning distance
The distance between two tabs: DeskProto will automatically set tabs along each selected profile using this distance. Here again you can either use a percentage (of the cuter diameter) or a fixed value.

Auto positioning distance overrides
In some cases one auto positioning distance is not enough, for instance when the operation contains both long profiles and short profiles. DeskProto allows you to set a **Minimum number** of tabs per profile to make sure that even the shortest profile gets (for instance) two tabs, and/or to set a **Maximum number** to prevent a long profile to get to many tabs.

The defaults for the above parameters can be set in the Default Vector Operation parameters (Options menu): on tab Profiling a button "Set..." is present to set the default values for support tabs. That button will open the Default Profiling tabs dialog.

Generate automatically
This button will make DeskProto remove all current tabs and set a complete series of tabs based on the parameters that you selected.

4.3.12 Default Profiling tabs



The use of Profiling tabs is explained on the help page for dialog Set Profiling tabs.
In that dialog they can be set manually or be generated automatically using the auto positioning parameters. offers.

It is also possible to set **Default profiling tabs** (on the Profiling page of the Vector operation parameters). To generate default tabs DeskProto will also use a set of tab parameters. This dialog Default Profiling tabs allows you to set the parameters for these default support tabs.
It can be accessed via the Default Vector Operation parameters

The parameters are the same as in dialog Set Profiling tabs and you can find all relevant information on that Help page.

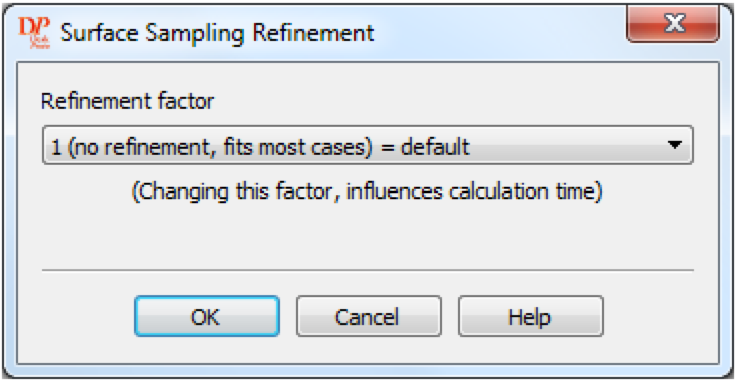
These same default parameters are the default values when opening dialog Set Profiling tabs.

4.3.13 Surface Sampling Refinement

The DeskProto toolpath calculations for Geometry data are based on a Z-grid, and the size of the grid cells sets the precision of the toolpaths. For every cell in the grid a Z-value will be calculated for the geometry (surface) at that location: the surface is sampled. In other words the DeskProto algorithm is based on **Surface Sampling**.

In the Geometry Operation parameters you enter the Precision as "Distance between toolpaths" and "Stepsize along toolpath". The smallest of the two will be used as Gridsize (cell size) of the Z-grid. While in most cases this works OK, sometimes it is needed to overrule this default and use a smaller Gridsize. This is called subsampling or sampling refinement, as per toolpath point more than one geometry position will be sampled. This dialog can be used to define the **Refinement factor**.

You can reach this dialog by using the button Settings on the Strategy tab of the Operation parameter dialog (not available for all strategies).

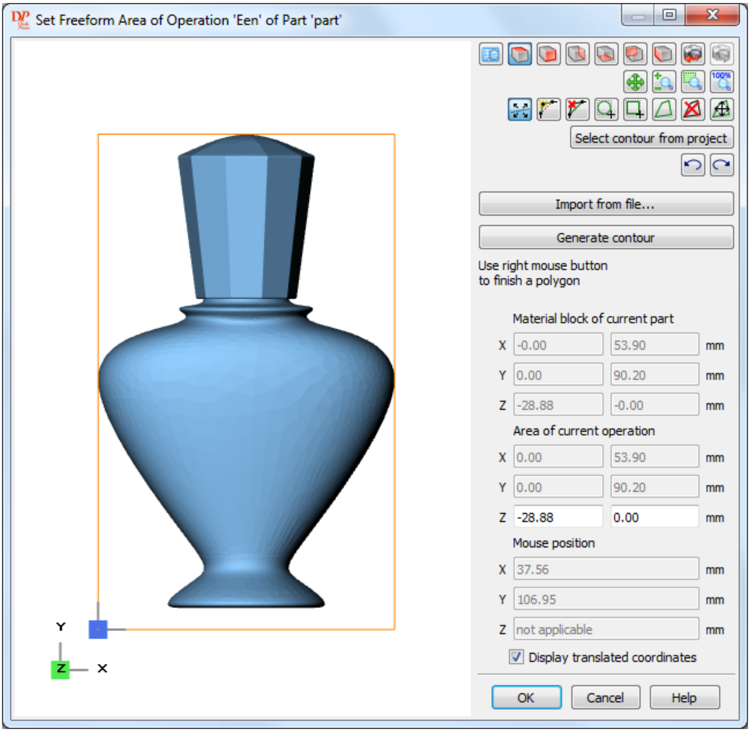


The Surface Sampling Refinement dialog allows you to fine-tune the gridsize or calculation precision. The higher the Refinement Factor, the more accurate the toolpaths and the longer the calculation time (quadratic). A refinement factor 1 will result in a Z-grid with a gridsize that equals the settings for Distance between toolpaths and Stepsize along toolpath. A **Factor 2** will double the Z-grid resolution for both X and Y, so will subdivide each grid cell into 4 smaller cells, etc.

Note 1: This setting is in fact only meant for advanced users, to be applied in special circumstances only !!
Note that higher factors will mean much longer calculation times !

Note 2: The default factor as said is 1 (None). However, for strategies Waterline and Contour Only the default factor is 3 as this is needed to achieve smooth toolpaths. This explains why the calculation times are longer for those strategies.

4.3.14 Set Freeform Area



A Freeform area is meant to save machining time by exactly defining the area that needs to be machined. This area can be defined in this dialog, by creating a closed (freeform) contour line in the Top view of the part. In the image above one of the visible items is a orange rectangle: this shows the Material Block. The Freeform area needs to be completely inside this rectangle. For this contour you can also set a Min and a Max Z-value, the result is called a Freeform area. You can reach this dialog by using the button “Set Freeform” on the Area tab of the Operation parameters dialog.

On the right side of the drawing four rows of buttons are present.

The first button will open dialog Items visible for Set Graphically, in which you can select which items should and should not be included in this drawing.
The use of the other buttons on the first row will be known from the Toolbar.

The four buttons on the second row will also be known from the Toolbar. Mouse rotation is not present: this dialog only used the six main views. Together with the third and fourth row these are the *mouse function buttons*, of which just one is active (pressed).

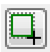
The eight buttons on the third row are new, these are the drawing tools used to graphically draw and/or change the area. These are *mouse function buttons* as well: of these mouse function buttons (13 in total) only one is active at any time. You will see that the shape of the cursor changes when you select a different mouse function button.

Adjust Boundary of a Freeform contour: you can move the points and the sides of the contour by dragging with the mouse. Watch the cursor to see if you will drag a point or a line.

Add new point to Freeform contour: click with your mouse to add a point. The closest side of the closest polyline will be split up into two new sides.

Delete point from Freeform contour: click with your mouse to delete a point. Watch the cursor when moving the mouse: when a minus sign shows you are on target.

Draw an Ellipse as new Freeform contour: click the left mouse button, move the mouse and release. The ellipse (can be a circle too) that is drawn is in fact a polyline. When you press the shift button during this input DeskProto will lock horizontal and vertical 1:1, forcing a circle.



Draw a Rectangle as new Freeform contour: click the left mouse button, move the mouse and release. This function is in fact also present in the rectangular area dialog. When you press the shift button during this input DeskProto will lock horizontal and vertical 1:1, forcing a square.

Draw a Polyline as new Freeform contour: each mouse click will add one point. The polyline is always closed. End the function with a right mouse-click.

Delete a Freeform contour.

 **Move** a Freeform contour: simply pick and drag the complete contour line.

These buttons are only active in the Top view and the Bottom view. In the other views only the Min Z and the Max Z level can be graphically set. You can also set the area's Z-values by entering them in the Z edit boxes. The boundaries of the current part are shown to assist you, so are the coordinate values of the current mouse position.

Button **Select contour from project** is available only when in the project one or more Vector files have been loaded, and when at least one closed vector curve is present. When you check this option you will see that the vector curves in this project are added to the drawing, drawn in gray. Light gray for open curves (cannot be used here), and dark gray for closed curves. Position the cursor on such closed curve and it will turn purple, click to select and it becomes black. Selecting more than one curve is possible by keeping the Shift key (on the keyboard) pressed while selecting.
This Select button is the thirteenth *mouse function button*.

The buttons on the fifth row are Undo and Redo. The freeform area dialog is the only place in DeskProto that offers Undo functionality, as while drawing this functionality is more needed than when working with settings in dialogs.

 **Undo** the last action using any of the drawing buttons.

 **Redo** the last action that was undone

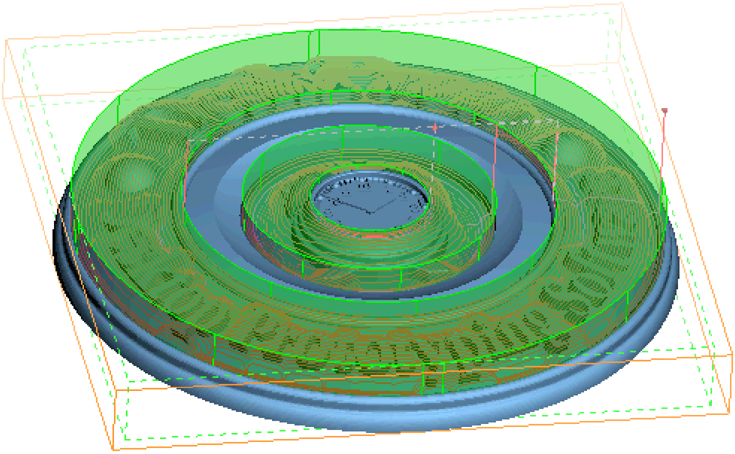
Nine levels of Undo are supported plus one level of Redo, so a total of 10 situations is stored.

The freeform area is defined by its contour, which is a 2D vector curve. Using the buttons described above your can draw a new contour. Other options are to use an existing curve and to make DeskProto generate a curve.

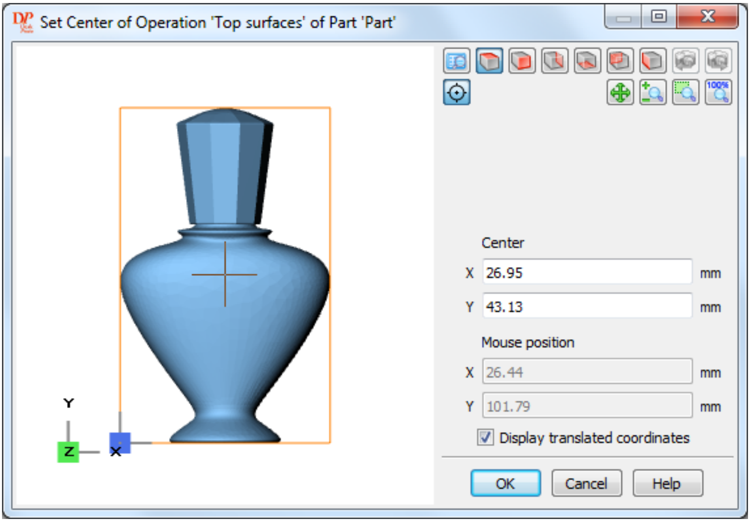
This contour line to define the freeform area can be imported from a [2D DXF file](). The button **Import from File ...** will open a standard File Open dialog to browse the correct file. This can be very handy in case you can export your 3D geometry and a 2D contour from the same CAD program.

A very powerful option is offered by button **Generate contour**: DeskProto then will automatically generate a freeform area contour that follows the outside contour of the current geometry. This will also work for multiple geometries, and for geometries that contain holes.

The Freeform area supports **multiple contours**: so you can draw more than one freeform area in this dialog.
Nested areas are supported as well: The outer contour defines an area to be machined, the inner contour defines an area to be skipped. And inside the inner contour you can again draw a new outer contour, and so on. See the illustration below.



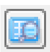
4.3.15 Set Center



In order to calculate toolpaths in a circular, spiral or radial toolpath a Center point is needed. For these strategies the center point can be set in the Operation parameters > Strategy tab > Detail settings: here you can enter the X and Y coordinates for this center point. At that same location you can also press the button **Set** in order to open this dialog to graphically set the center. The center point may be located outside the area to be machined, even outside the material block.

The dialog is in fact almost identical to the other dialogs for graphical input: Set Area and Set Freeform Area.
Only the functionality is different, and is in fact very limited:

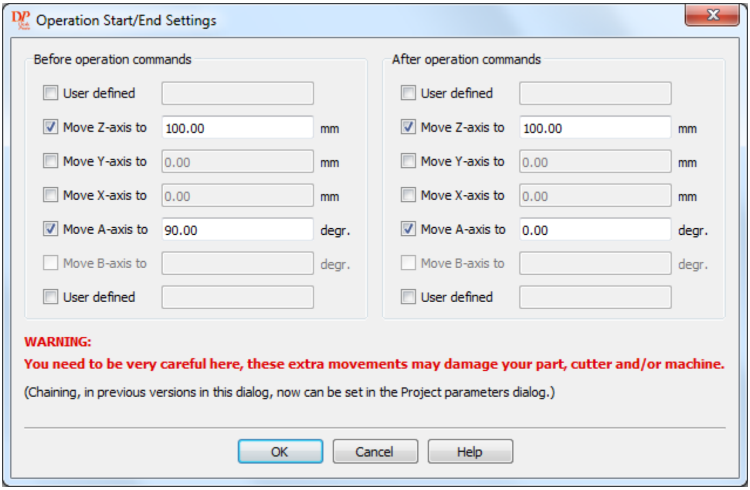
Mouse function Set Center allows you to click a point to set the XY coordinates for the center point. The large + sign in the drawing shows the current location of the center point. This only works in the Top View and in the Bottom View: in any of the other views only one of both coordinates is changed when clicking.



The first button will open dialog Items visible for Set Graphically, in which you can select which items should and should not be included in this drawing.

The use of the other buttons will be known from the Toolbar.

4.3.16 Operation Start/End Settings



This dialog is meant for advanced users only, as you can conclude from the warning in red text.
It enables you to add extra command lines to your NC program file, both at the Start (so just before the calculated toolpath of this operation starts) and at the End. For instance to let the cutter travel to some safe position after finishing the operation. Or to let the A-axis rotate to a certain angle before starting to machine the operation.

You can reach this dialog by pressing the button "Settings" for start/end on the Advanced tab of the Operation parameters dialog, for all three types of operations (Vector, Geometry and Bitmap).

Up to seven **Before Operation commands** (or Start commands) can be specified. These will be written as extra lines in the NC program file, just before the toolpath of this operation starts. Each line is optional, and will only be written when checked (so checking none will mean that no Start commands are inserted) :

- **User defined** can be used to issue **any** command, like for coolant or for some other device. Note that the line will be written exactly as defined in this edit box: so take care what you enter ! You need to know the language

that your machine requires: *this line will not be translated by the postprocessor*.
We advise that this option be used only by advanced users !!

- **Move Z-axis to** adds a Z-movement command to the NC file: to the specified position (in mm or inches) in workpiece coordinates. Note that these lines are written to the file in the same sequence as present in the dialog: first the Z-movement, and then the X, Y and A movement.
- **Move Y-axis to** adds a Y-movement command to the NC file: to the specified position (in mm or inches) in workpiece coordinates.
- **Move X-axis to** adds a X-movement command to the NC file: to the specified position (in mm or inches) in workpiece coordinates.
- **Move A-axis to** adds a command to rotate the A rotation axis to Angle degrees.
- **Move B-axis to** adds a command to rotate the B rotation axis to Angle degrees.
- **User defined 2**, see the explanation given above.

Note: the five Move-to commands are not available for some machines. A machine that always needs three coordinates per command without specifying which is for X, Y or Z cannot accept a line with for instance only a Z-axis coordinate: it won't know for which axis it is meant. For such machines all Move commands will be grayed out.

Note: The A-axis and B-axis commands are available only when the machine for this part has such rotation axis defined in its machine-definition, if not it will be grayed out. So the illustration above is made with a four-axis machine selected: the B-axis is not available.

These two rotation commands are used for **indexed machining**: machining a part from several sides, using three-axis machining for each side, with a rotation in-between. For instance the N-sided milling wizard uses the A-axis start command for this aim: milling from four sides will involve four operations, each time with a 90 degree rotation in-between. It is advised to also use a Z-axis command in that situation: to make the cutter move upward before the A-axis starts rotating.

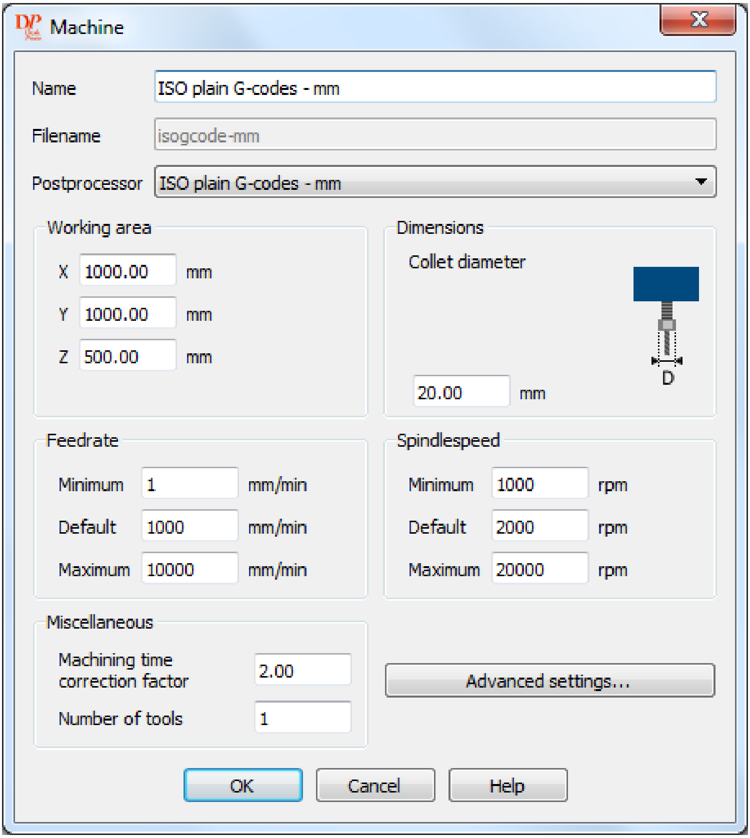
The seven **After Operation commands** are the same as the Before commands just described, only these will be written to the NC file after the toolpath of this operation.

In the four edit boxes for User defined an extra feature is hidden: you can enter new lines when needed, to create a **multi-line command**. Entering the string “^\N” will start a new line.

An alternative use for the User defined fields is to write one (or more) **comment lines** with information for the machinist. For instance telling which cutter needs to be loaded.

4.4 Libraries

4.4.1 Machine



The dimensions and speeds in the screenshot above are for a "generic" machine, to be used for any machine that runs on ISO G-codes. When you use this machine definition you will need to change most of this values to match the properties of your own machine.

The **Name** is the name that will appear in any DeskProto dialog for selecting a machine. It needs not be the same as the filename: use a name that clearly indicates which machine you mean. Each machine must have a unique name.

The **Filename** will be used to store the machine definition, using the file extension .MCH . When editing an existing machine you can no longer change the Filename. You can also add and remove machines by adding and removing MCH files to/from the DeskProto Drivers folder (as set in the Preferences).

The **Postprocessor** you select for this machine will be used to make the actual NC programs: see the Postprocessor library. The postprocessor is the most important setting in the machine definition, as it determines the format of each NC file to be written.

Most other values you enter in the Machine dialog are in fact less important as they will be used only to check whether the parameters entered later do not exceed the machines' capabilities. So these values do not influence the resulting toolpath (only the collet diameter does, and some of the Advanced settings).

The **Working area** is used for validation, to see if all the toolpaths will fit within the reach of the machine. Also, when drawing the working area of the machine a box of these dimensions is drawn.

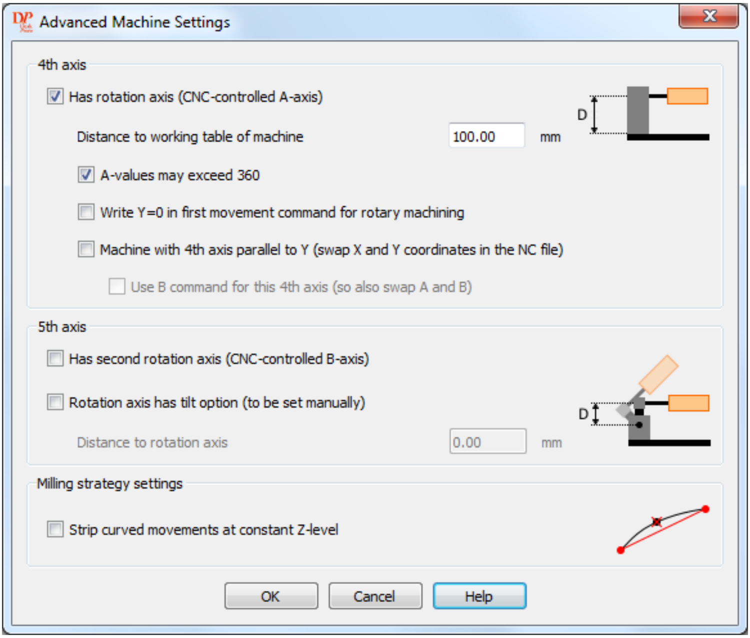
The **Collet diameter** as defined at the Dimensions section is used for the Collet collision check: to prevent the collet from damaging the model.

The **Feedrate** and **Spindle speed** Min and Max values set here are used to validate the speed settings in the operation parameters. The default values are used when selecting a different machine for a project: the speed values for all its operations then will be set to these defaults. In case your postprocessor uses a feedrate command for rapid movements, that rapid feedrate is determined by the maximum feedrate that is set here.

The **Machining time correction factor** is exactly what its name suggests: a factor that is used for this machine to multiply the theoretical machining time with in order to get an estimated real Machining time. This factor has to be a value larger than 1.
The **Number of tool**s is the number of tools that this machine can store and select automatically. Only machines that have an automatic toolchanger should use this. This option is also used for validation only.

The **Advanced settings** button leads to the Advanced settings.

4.4.2 Advanced Machine Settings



This dialog is a part of the Machine definition of DeskProto, and can be reached via the Advanced Settings button on the Machine definition dialog. The advanced settings configure the availability and dimensions of the optional A and B rotation axes.

4th axis:

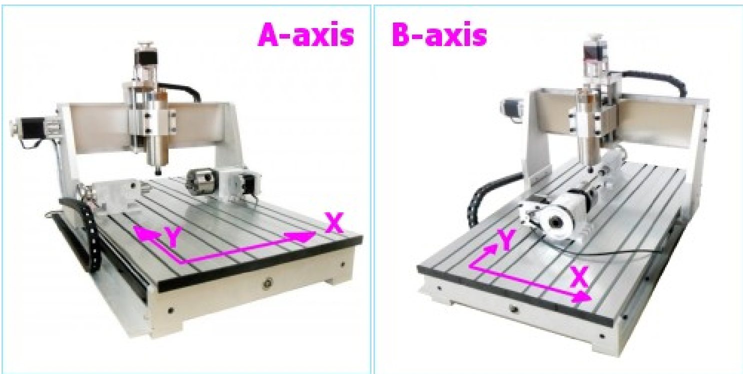
The A-axis is a device that rotates the part during machining, meaning that you can machine from all sides. Image it like a roast that is rotating above a barbecue. Such 4th axis is a very common option on CNC milling machines.

The option **Has rotation axis** needs to be checked in order to make rotation axis machining available for this machine. If not, the option "Use rotation-axis" in the Part parameters will be grayed out, and in the Start/End commands no A-axis commands are present.

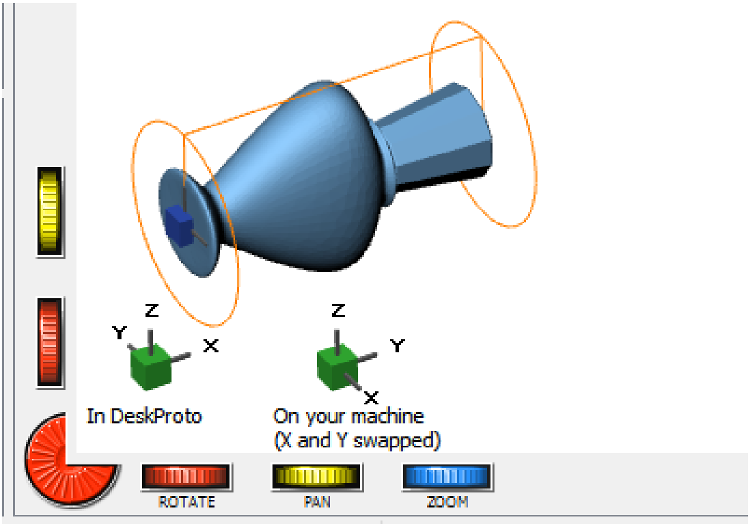
The **Distance to working table of machine** is the distance between the actual rotation axis center line and the machine table below: this value determines the maximum block diameter (or in fact radius) that still can be rotated on this machine.

On many machines the rotation axis can only revolve a limited number of times, and has to then rewind. Either because of mechanical limitations or because of software limitations. Other machines allow you to keep rotating in one direction: you then can check the option **A-values may exceed 360** degrees. Note that the angle value that is sent to the machine will then keep growing (for instance after 100 rotations it will be A = 36000 degrees).

Rotary machining in DeskProto in fact only uses three axes: X, A and Z: in the NC file no Y-coordinate will be present. This means that before starting such NC file the cutter needs to be moved to Y=0: exactly above the rotation axis.
The option **Write Y=0 in first movement command for rotary machining** adds this movement to Y=0.0 to the NC file. The default is that this option is NOT checked, as the movement can be dangerous: it will be done at the current Z-level, and if the Z is too low it might drive the cutter straight into the block. So when using this option make sure that the cutter is positioned high enough to let the cutter move over the block.



The 4th axis in DeskProto by default is an A-axis, so a rotation axis that is parallel to the X-axis.
However, on many desktop machines the rotation axis is parallel to Y, as that will allow a longer rotation axis for these machines. You can use DeskProto on such machine by checking the option **Machine with 4th axis parallel to Y (swap X and Y coordinates in the NC file)**. Checking this option will add an extra Orientator cube to your drawing:



The two orientators match the two machines as shown above. The orientator on the left shows the situation in DeskProto, which an standard machines equals the situation on the machine. When the X and Y coordinates are swapped the NC file will be conform the orientator on the right: with the rotation axis parallel to Y. These orientators are visible only when the item has been checked in the Items visible dialog.

Formally speaking such rotation axis parallel to Y is called a B-axis. However, many machine suppliers still use the A coordinate value to control it (as it is the 4th axis). When you have checked the option to Swap X and Y coordinates DeskProto as default will keep writing A-coordinates. Checking **Use B command for this 4th axis (so also swap A and B)** will make DeskProto use B-coordinate values instead.

For the **5th axis** DeskProto supports two different types.

The **first** type of 5th axis is a CNC controlled B-axis: so an axis identical to the 4th axis, but then parallel to Y instead of parallel to X. You can enable such 5th axis in DeskProto by checking the option **Has second rotation axis (CNC-controlled B-axis)**. When both the 4th axis and this 5th axis are configured the result is a five-axis milling machine. DeskProto only supports five-axis machines where the part rotates (so not machines where the cutter rotates) that have a trunnion style configuration.

The photo above shows such **trunnion style** machine: the two rotation axis units are built on top of one another. On the photo is seems that the small rotation axis is parallel to Z. That can be easily fixed though: when the large rotation axis turn 90 degrees the result will be one axis parallel to X and one parallel to Y. In DeskProto the workpiece zero point needs to be set on the exact location when the two rotation axes intersect.
In DeskProto this 5th axis can be used for indexed machining, using Start commands in the Operation parameters.

The **second** type of 5th axis is absolutely not common: a **Rotation axis tilt option**. Only a few machines support this (the Roland JWX-10 and MDX-40). As the icon on the dialog shows the complete 4th axis unit will be tilted, like a draw-bridge being opened. As this rotation is around the Y-axis, technically speaking this is a B-axis rotation. The advantage is that when machining a ring this rotation allows you to also machine the inside of the ring. Note that DeskProto only supports **manually controlled** tilt option.
For more information and a photo see the Tilt option tab of the Operation parameters.
If this option is not checked, the option "Use rotation-axis tilt option" in the Part parameters will be grayed out.

One extra parameter is available for this option: the **Distance to rotation axis** which sets a vertical distance between the A-axis and the B-axis (see the illustration). For the machines with this tilt option that we know this distance is 0.0 however other machines might exist that need such distance.

The third advanced machine setting is an optimization for the toolpaths: **Strip curved movements at constant Z-level**. This optimization makes the NC file shorter by deleting some intermediate points on the toolpath.
Each toolpath is built using a large number of short straight lines, the length on each line-segment determined by the Stepsize along toolpath. A long straight toolpath will also have been calculated as a series of segments, and for such straight movement in fact all intermediate points can be skipped: one long line will result in the same cutter movement. This optimization for straight lines is done automatically in DeskProto.
This easy approach is not possible for curved toolpaths like Circular and Spiral, as DeskProto does not support arc movements. A circular toolpath also is built using a large number of short straight lines, however here deleting a point slightly changes the toolpath. Still when machining a flat horizontal surface such small changes do not matter at all.

For some machines deleting points will make the movement faster and also smoother, as the controller of that machine does not have enough calculation power to calculate each small movement in time to keep the machine on speed. The result will be a non-smooth, slow movement. For such machine you may check this option, then DeskProto will delete half of the points on the circular toolpath. The result will be a faster and smoother movement of the cutter.

Note that for some other machines checking this option will have a contrary result and make the movement slower. This will happen for machines with a very fast controller, that checks the angle between two consecutive line segments on the toolpath and only keeps up speed in case these segments are almost parallel. Deleting intermediate points will increase the angle between the remaining line segments and force the machine to slow down. So for these machines you should not check this option.

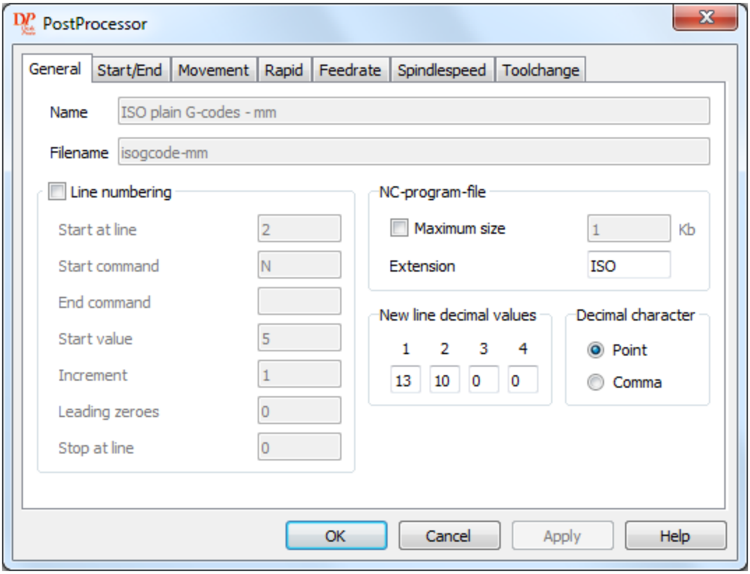
4.4.3 Postprocessor

The postprocessor determines the format of each NC file to be written. Compare this with a printer driver: it translates the contents of a print file to the language that the printer understands. Same for the postprocessor: each

machine manufacturer uses his own language for NC files (even when an ISO standard is followed), and the postprocessor translates to that language.
This dialog allows you to configure a postprocessor to write exactly the language that your machine requires.

The postprocessor dialog is divided into 7 tab pages.

General settings



The **Name** is the name that will appear in any DeskProto dialog for selecting a postprocessor. It needs not be the same as the filename: use a name that clearly indicates which postprocessor you mean. Each postprocessor must have a unique name.

The **Filename** will be used to store the postprocessor definition, using the file extension .PPR . When editing an existing postprocessor you can no longer change the Filename. You can also add and remove postprocessors by adding and removing PPR files to/from the DeskProto Drivers folder (as set in the Preferences).

When the output file needs **Line-numbering**, you can switch it on the checkbox. In this group you can define:

- **Start at line** - first line in the NC file with a line number
- **Start command** - command to be written before the actual line number (normally an "N")
- **End command** - command to be written after the actual line number (normally empty for nothing)
- **Start value** - the first number to be issued
- **Increment** - the difference between subsequent numbers (so an increment of 5 will result in line numbers 1, 6, 11 etc).
- **Leading zeroes** sets the number of positions to be used for the line-number: a value 0 (default) will output the number only, like in N123, a value of 5 will make DeskProto use 5 positions, resulting in N00123.
- Finally the option **Stop at line** makes it possible to have one or more unnumbered lines at the end of each file. For a 0 entry all lines are numbered, for 1 the last line is without number, etc. Only un-numbered End command lines (see next tab page) are possible, so the Stop value may not be larger than the number of End-commands on the next tab page.

In **NC-program File**, setting a **Maximum size** is needed for some very old milling machines, like for instance a Deckel Dialog 4. These machines need to completely read the NC file before they can start, while at the same time they have a very limited internal memory (say 256 Kb). For such a machine the NC program file has to be split up into parts no larger than 200 Kb or so. After applying this option DeskProto will automatically split the NC-program file into a series of files, which will be named like this: name, name#2, name#3 etc.
The **Extension** (file-extension) is used for every NC program file that will be created using this postprocessor. It does not influence the contents of the file, just the file's name.

The values you enter at the **New line decimal values** group will be put behind every line of the output file. Default these are set to the values 13,10,0,0 which will do for almost any machine. The value 13 stands for carriage-return, the value 10 stands for line-feed, and the values which are 0 will not be used by the postprocessor (unless followed by a non-zero value).
Do not touch these values unless you are absolutely sure what your are doing !

The **Decimal Character** will be used for any real number in the NC program files written. For instance the X-coordinate value 3½ will be output either as 3.50 (decimal point) or as 3,50 (decimal comma).

Start / End Settings

This page contains two editing windows called **Start commands** and **End commands**. Here you can enter the lines that every NC-program must start with, and the lines it must end with. This may include things like turning on the spindle motor, setting the units to metric or imperial, and other global functions. Please look at an existing postprocessor for an example. Also looking in an existing NC program file that works OK on your machine is helpful here.

The screenshot above shows a number of start commands and end commands. That is not needed: for simple G-code in most cases one start command is present ("%") and one end command ("M30", which stands for Program end).

Movement settings

The Tab page Movement determines the format of all movement commands in your NC program, which will be 99.99 % of its contents.

Each movement (so each line in the NC file) is built by a Start command, some Coordinates and an End command. To see their effect: just look at the **Example** line to see what will happen when you make a change.
The **Start command** (in the screenshot above "G01") determines that this is a linear line interpolation command, the **Coordinates** determine the required end position of this (linear) movement.
Most machines do not require an **End command**.
The option **Only for first movement** makes the Movement command '*modal*': after being given once it stays valid until a different command is given. As a result the Start command will not be repeated every line.

Each of the Coordinate values for X, Y, Z (and A, B) may be configured separately. A and B are between brackets as these are not used for three-axis machines.
A **Start command** and an **End command** can be defined for each coordinate. For instance a character like X, Y or Z to determine the axis, or a comma to separate the coordinates. Do not confuse this start command for one coordinate with the Start command for the complete movement command. Same for the two types of end-command on this page.

Positions gives the minimum number of character positions to be used. So if units is set to 7 and the output is 3.000 two spaces will be added, making the result in this example 'X   3.000'.
Decimals gives the number of units to be written behind the decimal point (or comma), so this affects the precision of the output. This precision will already be applied when drawing the toolpaths on the DeskProto screen. As a default for mm 3 decimals are used, and for inch 4.
The output of DeskProto is either in mm or inches (whichever you configured). When your output should be in any other unit you can change the **Factor** for the X, Y, Z coordinates. For instance a Factor 1000 will output the coordinate in units of 1/1000 mm or inch. The factor can also be used to reverse the direction of the axis: by entering -1.00 instead of 1.00

Last option for the coordinates is the **Default**: that value will be used when a coordinate is *Required* (see below) however when DeskProto does not yet know a coordinate value. A Default field may be either empty (used for most machines), a number (which will be interpreted as coordinate value, written to the NC file conform the specifications in the column above), or a string enclosed with quotes (which will be literally copied to the NC file, for instance "X6p", for Datron machines). So note that 0 and "0" will be treated differently.

Write only if changed means that coordinate values are only written in case changed. So when only Y has been changed DeskProto may write a command like "G1 Y20.0", meaning that the values for X and Z will remain the same for this movement.
Also sign positive values adds a "+" in front of every positive coordinate.
Skip trailing zeros makes the file's size smaller, by removing any insignificant zero in a coordinate value. For instance 3.400 becomes 3.4 and 3.000 becomes 3. Again please use the example line to see what will happen.

Using the **Order/Required** edit field you can change the order of the X, Y, Z, A and B coordinates in the output. Each of these five characters should be present in this edit box, and all exactly one time.
In this same field you can make a coordinate value **Required**, by adding a + in front of the character. So when you enter +X+Y+Z+AB this means that X, Y, Z and A are required. This is needed for machines where the position (sequence) on the line indicates the axis for which each coordinate is meant.
For instance "*G1 X10 Z10 A 180*" on such machine without the required field would become "*MOVE 10, 10, 180*", which would send the Y to 10 and the Z to 180. With the required field the result is "*MOVE 10, , 10, 180*" (note the extra comma). What will be written for the missing coordinate value depends on the setting on line Default, as just discussed.

As **Distance unit** group you can choose between mm and inches for the coordinate values to be converted to. Make sure that your machine uses the same units: for some machines you need to explicitly give the command G 70 (inches) or G71 (mm). You can do so at the "Start commands' of the Start/End tab page. This postprocessor setting is independent for the Units setting in the DeskProto Preferences. This makes sure that inch workers still can use a metric machine, and the other way round.
The **Angle unit** is only used for rotation axis machining (degrees or radians), so it does not apply if your machine does not have a rotation axis.

The (A) and (B) columns and the Angle units are only used in case a 4th axis / 5th axis is both present and selected. For three axis machines you can just ignore these columns.

Rapid Settings

Rapid movements are used to save milling time by moving as fast as the machine can travel. These are used by DeskProto for positioning moves above the top of the material block. Rapid movements can be achieved either by using a special Rapid **Start command**, or by first changing the feedrate and then using the normal Movement command, at **the maximum feedrate of the machine**. This maximum feedrate value can be changed at the Machine dialog.
The option **Only for first movement** makes the Rapid command modal: after been given once it stays valid until a different command is given.

Feedrate settings:

Feedrate commands will only be output if the option **Use** is switched on.
The option **Write only if changed** will be on for most machines: if not the feedrate will be output on every movement line.

The effect of using the **Start-command** and **End-command** can best be seen by looking at the example line at the bottom of this tab page.
An invisible special option is available in the fields Start command and End command: you can enter new lines when needed to create a *multi-line*

Feedrate command. Entering the string “^\N” will start a new line, and entering the string “^\V” will again output the actual feedrate value.
For Instance: entering the string “ F ^\V^\N G” in the Start command edit box (the actual Feedrate value to be output is 500). will cause DeskProto to write a two-line Feedrate command as follows:
F 500
G 500
Positions stands for the minimum number of positions that are occupied by the value of the feedrate.
Decimals stands for the number of characters behind the point (or comma).
A **Factor** is needed when on the DeskProto screen and in the NC file different units are used. For instance a Factor 1000 will output a number that is 1000 times larger than what DeskProto shows. For instance 50000 instead of 50 mm/sec)

The **Method** setting is self-explanatory.
The settings of **Units** are only used to be able to show the correct units in the dialogs, whenever you have to enter a feedrate. DeskProto does not "understand" them, it just copies the number that is set in the Operation parameters to the NC program file.

The correct feedrate for rotation axis movements (so cutting movements that also include a rotation of the A-axis) is a difficult issue. In standard G-code the feedrate is define only as linear speed, so a good controller should use the linear feedrate that has been set and then calculate how fast the rotation axis needs to rotate in order to reach the prescribed speed. That is a difficult task though, as the length of a rotary movement depends on the distance to the rotation axis. Unfortunately most machines cannot make such calculation.

DeskProto offers two alternative methods for rotary feedrate:
Use inverse time feedrate for rotary movements - instead of the speed, specify the time needed for the total movement. DeskProto knows the length of the movement, so it is easy to calculate the total time to complete this movement at the prescribed feedrate. The machine then can set the correct speed for each of the moving axes. *This is the best option: so in case your controller supports inverse time feedrate select this option !*
Use angular feedrate for rotary movements - specify the rotary feedrate in degrees per time unit. This solves the feedrate issue for movements that only include an rotation axis movement, however for combined movements (rotation axis + linear axis) the problem will persist.

Only one of both options can be checked (or none). In case one of these options is checked in extra tab page will appear in the postprocessor dialog:

"Rotary feedrate". Both options - so both extra tab pages - will be explained below.

Inverse Time Feedrate settings

This tab page will only be visible when on tab page Feedrate the option "Use inverse time feedrate for rotary movements" has been checked. It concerns the feedrate for cutting movements that include a rotation of a rotary axis.

When setting the feedrate in linear/time units it is difficult to determine how fast the rotation axis needs to rotate, when using angular/time units it is difficult to determine how fast the linear axis needs to move. A good solution is to instead **define the time that may be used for the total movement**. The controller then can easily calculate how fast each of the axes needs to move.

The Time for each movement can easily be calculated as Distance / Linear feedrate
Important is that when roughing layers are applied the (linear) Distance to be machined will be measured at the Maximum Z of the current roughing layer. This is needed as the linear speed through the material at that Z-level will be (much) higher than for the tip of the cutter.
For historic reasons the parameter for this option is not the time but the **inverted time** (1 divided by the total time).

Inverse time is a modal command: it needs to be switched on and off.
The **Switch commands** can be defined here:
- the **On** command in standard G-code is G93.
When switched on every movement command needs to contain a Feedrate (or rather time) value.
- the **Off** command in standard G-code is G94 (in fact this is the command to set Linear feedrate).

For **Units** only the Time unit can be selected: either minutes or seconds.

The settings for **Command** and for **Method** are identical to those on the Feedrate page.

Angular Feedrate settings

This tab page will only be visible when on tab page Feedrate the option "Use angular feedrate for rotary movements" has been checked. It concerns the feedrate for cutting movements that include a rotation of a rotary axis.

Some machines require the feedrate for rotary movements (either for movements that only rotate the A-axis or for movements that combine a rotary move with a linear component) to be set in **Angle per time** units.
On this page you can set the **Units** to be used:
for the Angle either degree or radian
for the Time either millisecond, second, minute or hour.

After selecting this option DeskProto will convert the linear feedrate (set in the Operation parameters) to an angular feedrate. Important is that when roughing layers are applied the (linear) Distance to be machined will be measured at the Maximum Z of the current roughing layer. This is needed as the linear speed through the material at that Z-level will be (much) higher than for the tip of the cutter.

In case your machine also support the Inverse time feedrate (see above) that method is preferred, as the Angular feedrate method is not accurate for movements that combine a rotary and a linear component in one movement.

The settings for **Command** and for **Method** are identical to those on the Feedrate page.

Spindle speed settings

Spindle commands will only be output if the option **Use** is switched on.

The effect of using the **Start-command** and **End-command** can best be seen by looking at the example line at the bottom of this tab page.
An invisible special option is available in the fields Start command and End command: you can enter new lines when needed to create a *multi-line Spindle command*. Entering the string “^\N” will start a new line, and entering the string “^\V” will again output the actual spindlespeed value.
For Instance: entering the string “ S ^\V^\N R” in the Start command edit box (the actual Spindle speed value to be output is 8000). will cause DeskProto to write a two-line Feedrate command as follows:
S 8000

R 8000
Positions stands for the minimum number of positions that are occupied by the value of the feedrate.
Decimals stands for the number of characters behind the point (or comma).
A **Factor** is needed when on the DeskProto screen and in the NC file different units are used. For instance a Factor 1000 will output a number that is 1000 times larger than what DeskProto shows. For instance 50000 instead of 50 rpm)

The **Method** setting is self-explanatory.

The Units to be used cannot be set: for the Spindle speed the unit always is RPM (Rounds per Minute, of Revolutions per Minute).

Toolchange settings

DeskProto will create one NC program file for one part. However, for every operation a different cutter can be used, so in case you are using different cutters in one part (which would mean in one NC program) you have to define what should happen at a tool change. Select one of the three options.

Use change-command in NC program can be used in case the milling machine is equipped with an automatic tool change system (ATC = Automatic Tool Changer). To define the actual command you can use one, two or three lines, as some controllers require separate lines to select the next cutter and to actually load it. For each of these lines you can select whether or not the **Tool Nr** has to be present on the line. The Example line on the bottom of this dialog will show the result.

In the illustration above two lines are used, each mentioning the tool number (this is just an example command: actual commands will differ per machine). The first line ("T1 M06") is pretty standard G-code: T for the tool number to be loaded, and M6 to start the ATC operation. Note the space just before the “M6”. For many G-code machines this first line will be sufficient. On the second line "G43 H1"sets the tool length compensation for this next cutter, which for this machine apparently is not done automatically. "Z1.0" after G43 moves the cutter to Z=1.0 for a visual check of the tool compensation, and "M08" switches the Coolant on. As you can see you can configure the postprocessor to meet your own preferences.

The same special options ^\V and ^\N as just described for Spindle speed (scroll up to find it) are available here as well, so if needed you can use more than three lines.

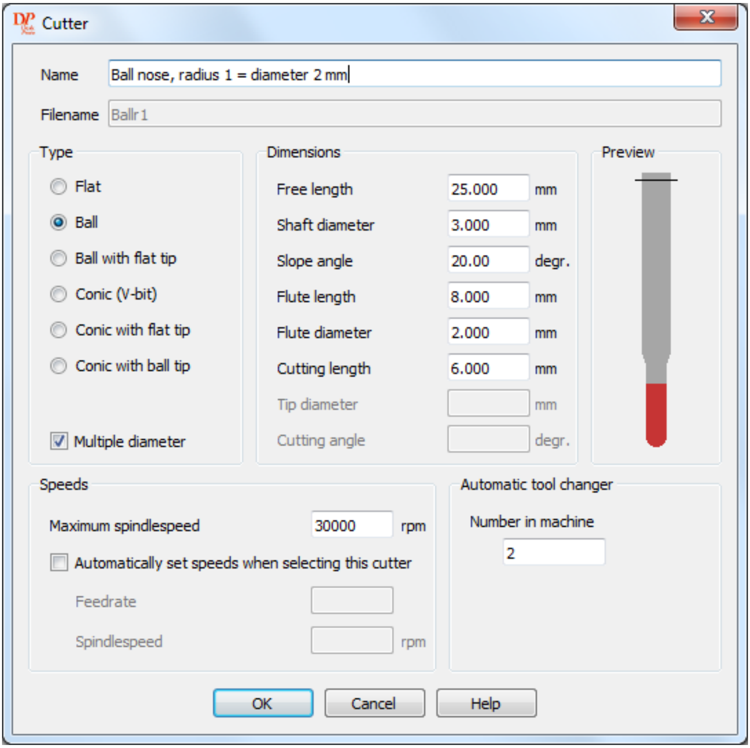
Note:
For this method be sure that the 'number of tools' of the machine is set correctly. You can do that in the Machine dialog. Also be sure that the cutters that you use in DeskProto have the correct "Number in machine" parameter. You can set that in the Cutter dialog. You also have to check if the correct cutter is indeed loaded on this position of the machine's tool changer: DeskProto just loads Tool No "N" without knowing if it is indeed the right cutter.

Use pause-command in NC program lets you change the cutter manually, however within the same NC program. The Pause command will stop the machine for this purpose. You must define the pause command in the pause field. For most machines such command is not available though. It needs to be a special Pause for toolchange command: a standard Pause will only wait for the prescribed time (which is dangerous when changing a tool) and will not allow tool length compensation (needed because the new cutter will have a different length).

Use new NC program will cause a next NC program to be created. If your machine does not support a tool change or if you do not know, it is best to choose new NC program file. After ending the first file you can then change the cutter, correct the zero position for the length of the new cutter, and start the second NC program file.
When writing such multiple NC program file, names for the subsequent files will be automatically generated by DeskProto. In case you chose the name Test.nc and the two operations were called Roughing and Finishing, then the first file will be called Test#1_Roughing.nc and the second file Test#2_Finishing.nc

Note:
Use new NC program is the safest, and thus it is the DeskProto default for this option

4.4.4 Cutter



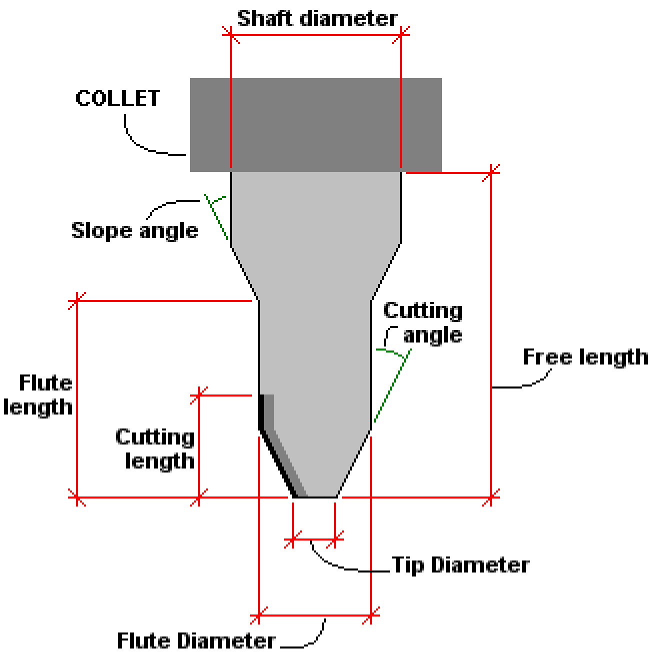
The **Name** is the name that will appear in any DeskProto dialog for selecting a cutter. It needs not be the same as the filename: use a name that clearly indicates which cutter you mean. Show in this Name whether you mean radius or diameter for any number (you will forget if 'Ball6' means R6 or D6). Each cutter must have a unique name.

The **Filename** will be used to store the cutter definition, using the file extension .CTR . When editing an existing cutter you can no longer change the Filename. You can also add and remove cutters by adding and removing CTR files to/from the DeskProto Drivers folder (as set in the Preferences).

DeskProto offers six **Type**s of cutter, that are variants of three basic types: flat (square cutter), conic (sharp point), and ball (ballnose cutter). Both conic and ball can be given a flat tool-tip, conic can be given a ballnose tooltip as well. Depending on the type that you select, some of the **Dimensions** might be disabled. For example a flat cutter does not have a cutting angle.

For each type you can check the option **Multiple diameter**, to define a tool of which the shaft (the part that fits in the collet chuck) is thicker than the flute (the part that actually cuts). This is a commonly used model for small cutters. When correctly defined, DeskProto will make sure that the thick shaft does not damage your model at vertical surfaces.

Just look at the drawing in the **Preview** to see the cutter that you have defined.
The image below gives an explanation of most terms used for the **Dimensions**.



The **Free length** is the length of the part of the tool below the collet. So this is not the total length of the cutter. The free length is not constant, as it depends on how far you insert the cutter in the collet. DeskProto uses this parameter only for the collet collision check. In the Preview image the free length is shown by a horizontal black line.

The meaning of **Shaft diameter** will be clear. For single diameter tools the Flute diameter is identical to this value. In order to support 1/8 inch (3.175 mm) cutters with a DeskProto set on metric, all dimensions in this dialog can be set with 3 decimals accuracy

The **Slope Angle** is available only for multiple diameter cutters. It defines the transition between the thick shaft and the thinner flute, which for most of these cutters has a conical shape.

The **Flute length** is only available for multiple diameter tools, as it is the length of the small diameter part which is called the flute of the cutter.

The **Flute diameter** (or flute diameter) is the nominal diameter of the cutter, used for the calculations (do not confuse Diameter with Radius!). It is only available for multiple diameter tools, as otherwise it is the same as the shaft diameter.

The **Cutting length** is the length of that part of the tool that actually cuts: it will be used to calculate Layers as the tool may not sink into the material deeper than this value. In case you do not select Roughing then DeskProto will automatically do so for the first Operation. In the Preview image the cutting length is shown by a red color.

The **Tip diameter** is the diameter of the flat tip for cutter types with a flat tip, and the diameter of the flat ballnose tip for Ball/Conic cutters. So for instance a Ball/Flat cutter with diameter 6 and tip 4 will have a radius 1 left (this type of cutter sometimes is called bull nose).
A special cutter type can be defined by choosing “Ball with flat tip” and setting the Tip diameter larger than the Flute diameter (which normally is nonsense). This creates a special cutter type “Curved tip” as a result. The preview drawing shows what will happen.

The **Cutting angle** is for conic cutters only: it is the angle between the cutting edge and the center-line of the tool (so the ‘grinding angle’). Do not confuse this with the ‘included angle’ that is also used. You can also use this parameter to define special tapered cutters having a draft angle of say 3 degrees.
Here as well: just try and look what happens in the Preview.

The **Preview** shows you the currently defined cutter-definition in a drawing, which is a very convenient help when setting the parameters in this dialog. A preview can of course only be drawn when a correct set of dimensions have been entered first (for invalid setting a red cross will be drawn). The area in dark red shows the cutting length, the horizontal line at top indicates the collet and thus shows the free length.

The **Number in machine** indicates on which position of the Automatic Tool Changer this cutter needs to be loaded. So this is used only when your machine supports automatic toolchange. The number will be used for any toolchange command written in the NC-program file.
Important:
This number can be different for each machine and even for each situation, as the operator has to load the correct cutter in the correct location.

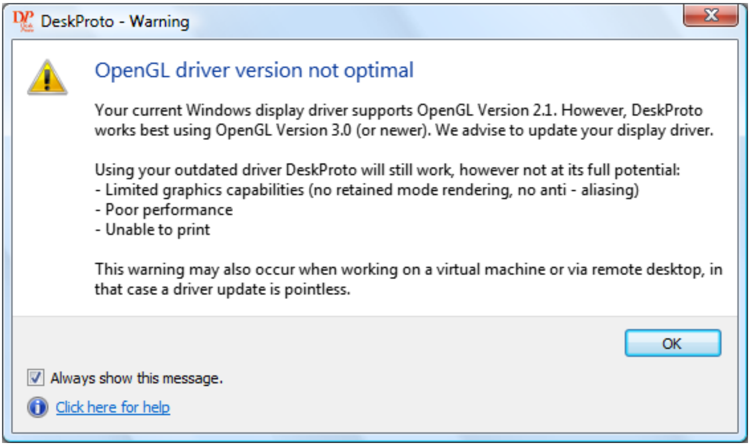
The **Maximum spindlespeed** is used only for validation of your projects (some larger cutters may not rotate at high rpm as these are insufficiently balanced).

The option **Automatically set speeds when selecting this cutter** may be very handy when you have standard milling conditions. For instance when you always machine in wax (jewelry wax models) you may want to always use the same low feedrate and high spindle speed for a certain thin cutter. Then you can define these two speeds here, and in the Operation parameters DeskProto will automatically set these values for **Feedrate** and **Spindlespeed** when this cutter is selected.

Note that for the Feedrate DeskProto will just copy the number that you enter in this dialog to the NC file, without understanding it's units. So when enter "10" and use this cutter on a m/min machine the feedrate will be 10 m/min, on a mm/sec machine it will be 10 mm/sec. A large difference ! DeskProto could show the units used by your current machine, however that would give a false sense of security as they would be valid for that machine only.

4.5 OpenGL warnings

4.5.1 OpenGL driver: version



DeskProto uses a graphics language called **OpenGL** to display graphics information (a drawing) in its View Window. This language has been defined long time ago, and over the years already many new Versions have been defined, each time adding extra functionality to the language. DeskProto needs OpenGL **Version 3** or newer, when your driver version is too low this warning will be shown when the program is started.

When this warning is give you can continue using DeskProto, however at some points the graphic behavior will be below standard. The advise is to download and install a new driver for your graphics card. You then need to find out the brand and type of your graphics card, visit the website of the card manufacturer and there look for a driver for your type card. In most cases you can download drivers in a section called Support or Download.

In case you have a choice, select the driver that supports OpenGL.
Like for all AMD Graphics cards (formerly ATI), where you can choose between downloading
1- the Catalyst Software Suite
2- the WDM Integrated Driver
Here you need to select the Catalyst suite, as the obsolete WDM driver (Windows Driver Model) does not support OpenGL. Windows then will fall

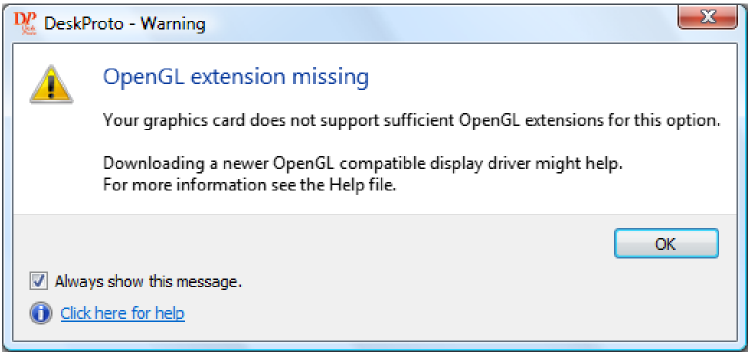
back on its old software renderer (OpenGL V1.0) to create the graphics, which misses some features and will also make DeskProto very slow.

Important to know is that using the button "Update Driver" in the Windows Device Manager does not always find the latest driver ! So you need to check the card manufacturer's website.

A big advantage of OpenGL is that it makes hardware implementation possible: let specialized hardware on the card do graphics calculations, making it much faster than using software. The more expensive your graphics card, the more can be done by hardware, and the faster all graphics will be displayed.
An important feature is the **Retained mode** rendering, as mentioned in the warning text. This means that the complete 3D scene is stored in the card's graphics memory, using a **'vertex buffer'**. So in order to change the viewpoint, the software only needs to send the new camera position to the card, the rest will all be done in hardware. You can imagine that this is much quicker than again sending the complete scene to the graphics card for each change (which is called **Immediate mode**).
Vertex buffers are available from OpenGL V 1.5

A few OpenGL related setting can be found in the Preferences. In case of a limited driver not all functions will be available.

4.5.2 OpenGL driver: extensions



DeskProto uses a graphics language called **OpenGL** to display graphics information (a drawing) in its View Window. This language has been defined long time ago, and over the years already many new Versions have been defined, each time adding extra functionality to the language. DeskProto needs OpenGL V3 or newer.

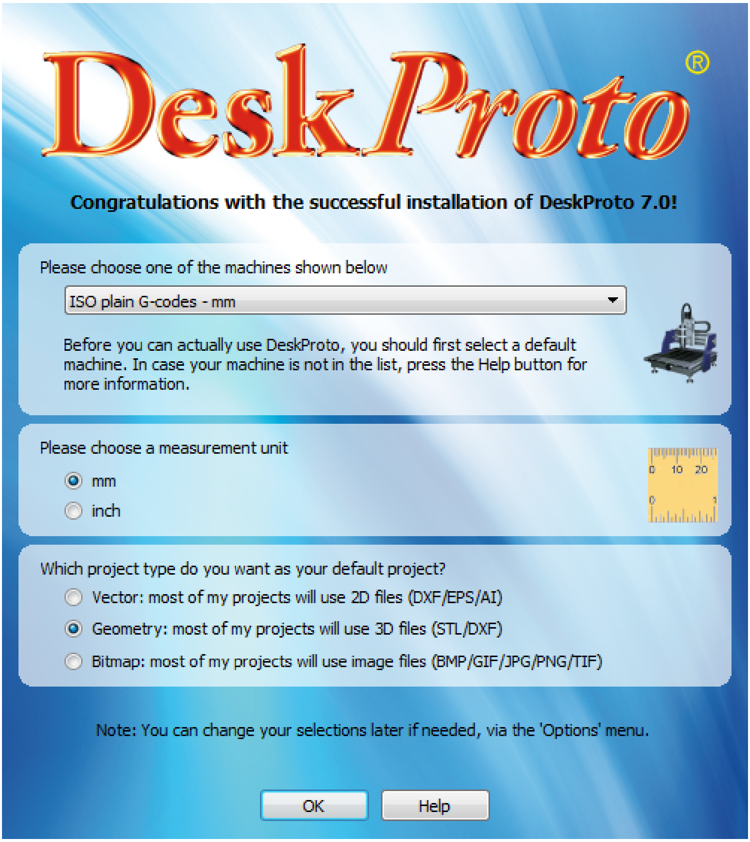
Apart from these differences between various versions the OpenGL language supports "**extensions**": extra functionality that is not required and thus may and may not be present.

A few features are present in DeskProto that depend on such extension: if not present the function cannot be executed and this warning dialog will be shown. This concerns the following features:
- Print Image (File menu)
- Show points (Items visible)
- Generate Contour (Set freeform area).

The above warning will be shown when one of these functions is called.

Some other features will simply be disabled (grayed out) when the required extension is missing.
For instance Anti-aliasing (Advanced Preferences).

4.6 Initial Personal Settings



This dialog will be shown only once: when DeskProto is started for the first time after it has been installed.

You have to select the **Machine** that will be used as the machine for every new project. Click on the arrow to show the list that you can choose from. When your machine is not listed: read further below.
DeskProto asks you to select a Machine, so not (like many other CAM programs) a Postprocessor. In DeskProto the postprocessor is one of the settings in the machine definition: when selecting a machine you also select a postprocessor. Advantage of this construction is that several machines can share the same postprocessor.

The second question asks to set the **Measurement unit** you want to use within DeskProto: mm or inch. All user-interaction will show this unit, and all CAD-data files will be loaded using this unit (because in most of the CAD-files the units used are not specified).

Third question is about the **Default project** that you want to configure.
DeskProto offers three different types of machining:
- Vector: load a 2D drawing file and make the cutter follow curves in a drawing
- Geometry: load a 3D geometry file and generate toolpaths to create that shape
- Bitmap: load a bitmap image, convert it to a 3D relief can generate toolpaths to machine it.
Selecting one option here tells DeskProto which type of project to open as default New project: the other two types remain available. Select the type of project that you expect to be using most.
This default type can be changed later by editing the Default Part (Options menu) and changing the type of operation that is present.

What to do when your machine is not listed ?
This is a very common situation (as the number of machine builders and the models that they produce is almost infinite), and in most cases it is easy to solve. Please follow the three instructions given below.

1. Look if you can find a **different model** made by the same machine builder. In most cases a machine builder uses the same NC file format for all machines, so you may expect that the resulting NC files will be OK for your machine as well. Try to run an NC file, and if positive proceed to "Add machine" below.
Important: when you look for your machine in the alphabetical list you also need to look under the *Manufacturer's name*. For instance the Magic7 machine is listed as "REDT - MAGIC-7", and the High-Z machines are listed as "Heiz High-Z"

2. Look if you can find a **generic machine definition** for the controller or control software that you use. Like for the Mach3 control software one of the machines called "Mach3/Mach4 based machine", or for the EdingCNC controller the machine called "EdingCNC based machine". Try to run an NC file, and if positive proceed to "Add machine" below.

3. Most machines will run on NC files written in **standard G-codes**. G-code is the official standard for NC files, defined by ISO (the International Standards Organization). Unfortunately each machine builder has developed his own flavor of this standard, and not all G-code files are compatible. Still when your machine runs on G-code, you have a good chance that it will work when you select the machine called "ISO plain G-codes". Try to run an NC file, and if positive proceed to "Add machine" below.

In case the above three options do not work and you have problems configuring a machine definition and a postprocessor, feel free to send us an email: we can assist ! We are always happy when we can add a machine to the list this way.

Add machine. When a test is positive then you can add your own machine to the list as follows:
Options > Library of machines > OK on warning > Select the machine that you tested with and press button **Copy**. Now in the Machine dialog give the new copy a proper name and filename. Make sure that you do not change the postprocessor. Enter the correct dimensions and speeds, if needed add a rotation axis in the Advanced machine settings. For more information use the Help button of the Machine dialog.

When your machine has an optional rotation axis it is possible that the machine-definition in DeskProto does not have that option checked. It is easy to add it in the machine definition (see the previous paragraph), on the Advanced settings page.

The default Machine is stored in the Default Project, and can be changed later by choosing the option Default Project Parameters from the Options menu.
The measurement unit will be saved in the Preferences. You can later change this setting in the Preferences dialog (Options menu).
The default project type as said can be changed by editing the Default Part (Options menu) and changing the type of operation that is present.

These initial settings will be stored per user of the computer, so any new user will see this dialog when he/she starts DeskProto for the first time.

4.7 Edition select

DeskProto is offered to you as **Freemium** software:
you are welcome to use the *basic functionality* DeskProto **free** of charge, the *advanced features* are available as **premium** extras.

Four different Editions are available:

- **Free Edition**
- **Entry Edition**
- **Expert Edition**
- **Multi-Axis Edition**

of which the first is free while for the other three you need to buy a license.
The Free edition allows you to also **Trial** (evaluate) the higher editions: when running in **trial mode** the resulting toolpath will leave a **Trial cross** (watermark) on each part that is machined.

The **Free Edition** is available for anyone: free of charge, without any obligations attached. It's functionality is limited, still it offers all you need for

basic CNC machining: Profiling toolpaths based on Vector Data, Parallel toolpaths over Geometry Data, and machining reliefs based on Bitmap data.
In the Free edition a project may contain maximum one part and one operation.
Many parameters as described in the Help file are not available in the Free Edition. Still the most important parameters are there, and for many users this free CAM program will be all they need.

The **Entry Edition** is the lowcost version of DeskProto, offering limited options, at a very low price.
A few important extra options are present (added to the Free edition): Pocketing and Drilling for Vector operations, and Roughing for all three operation types. Projects also may contain any number of parts and operations.

The **Expert Edition** includes all parameters, except for the rotation axis options. So the fourth axis and the fifth axis are not available in this edition.

The **Multi-Axis Edition** is the most complete version: all parameters are present, and so are the A-axis and B-axes rotation axes.

More information about higher editions can be found in the Upgrade dialog.
An edition comparison table can be found on www.deskproto.com

A license for a higher edition can be activated in the Activate dialog.
After activation your license this Edition select dialog will no longer be displayed.

4.8 Start Screen

The DeskProto Start Screen shown above is a help to quickly start the task that you need.
You can open the Start Screen via the File menu (File >> Show Start Screen).
A keyboard shortcut is available as well: Ctrl + B (B for "Begin", as the S already is used).

It will first show the Edition that your are running,
next your Name and Location (as owner of the license),
and the type of License, with the restrictions that apply (of any).

Three groups of tasks are presented:
- Open recent project
- Start new project
- Other tasks

Each line in these three lists is a link that will directly start that task.

Recent projects will be available only when you have used DeskProto V7 before. In this dialog only the latest 5 projects will be shown. Command Open will allow you to open any project file by browsing one.

Starting a New project can be done either using the Wizard interface or the Dialog based interface.
Default choice for this dialog is to start a New project using the Wizard: that is what will happen when you simply press the Enter key (indicated by the icon at this line). This is the most convenient option for novice users.
The difference between Vector/Geometry/Bitmap projects is explained on page New project.

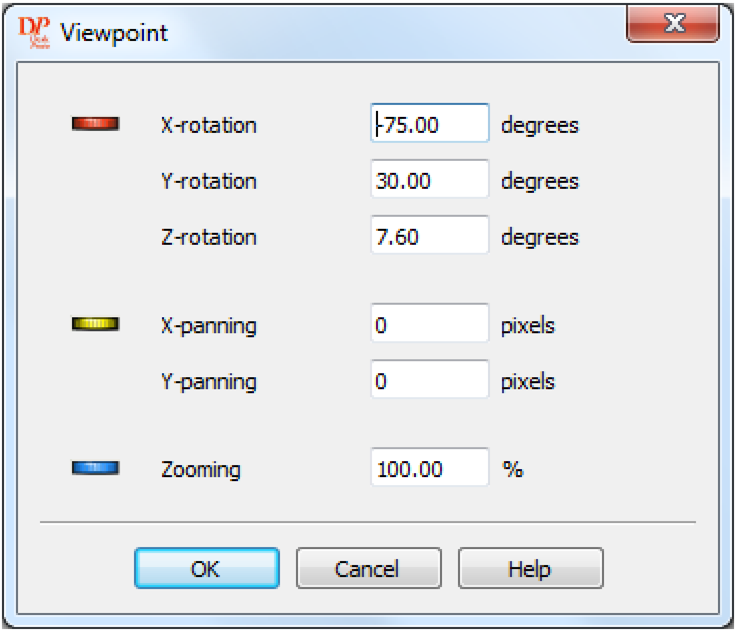
The DeskProto Setup has installed a number of **Sample projects** and **Sample geometries** on your PC. Including some great geometries like the DeskProto picture frame: see the lessons in the DeskProto Tutorial book.

Conform Microsoft's specifications the **Samples** have been installed in the \ProgramData\ folder, which may not be easy to find as it is a hidden folder and its location is different per Windows version. Sorry about that, unfortunately for standard users other locations are not permitted by Microsoft.
Using the checkbox **Use samples folder** you will always be able to find the sample geometries.

In the **Other tasks** section, **Open sample project** makes it easy to find the sample projects.
The other three 'other tasks' require an Internet connection, as they will start your browser and open a web page.

The checkbox **Show this start screen on startup** makes this screen automatically appear at each start of DeskProto. After having de-selected this option, you can still access the Start Screen using the command **Show Start screen** in the File menu.

4.9 Viewpoint



The View menu command Viewpoint -> Custom... displays the **Viewpoint dialog**, in which you can change the settings of the active view, that is the settings for rotation, panning and zooming. In fact this is setting the Camera position.

Shortcuts:

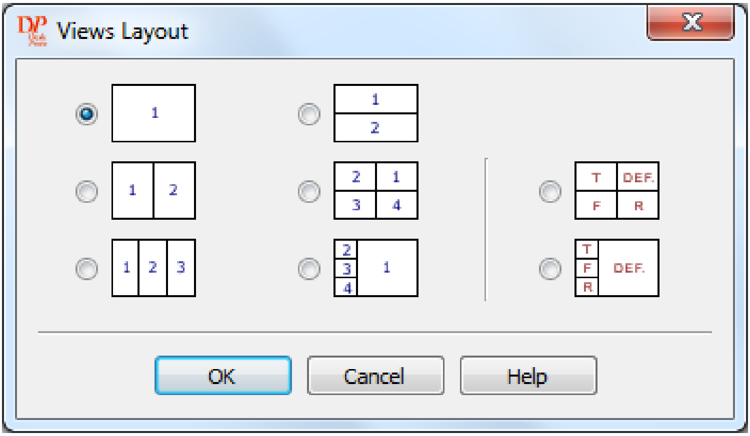
Toolbar: 

Mouse: Right-click in a view, and select the Viewpoint item in the shown context-menu.

In this dialog you can change the point of view from which you look at the curves, the geometry, z-grid, toolpath etc. You can set the values exactly, using the keyboard. The rotations are executed in the order X, Y, Z. The effect is the same as when using mouse rotation, pan and zoom or when using the thumbwheels.

Note:
These values do not change the toolpaths, they only change the Camera position. To rotate/pan/scale the part to be machined you should go to the Transform tab page of the Part Parameters dialog.

4.10 Views Layout

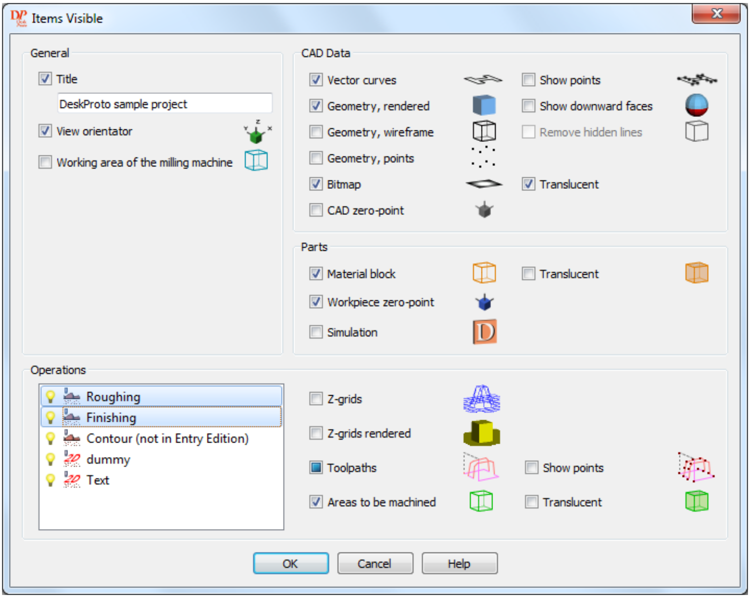


In this dialog you can change the Layout of the View Window: the way it is filled with Views. It is possible to show 1, 2, 3 or 4 Views at the same time, as shown in the dialog. Each of these Views can have different settings for Viewpoint and Items visible. All these settings will be stored in the project file.

The two layouts at the right automatically set the viewpoint for all views. Those are the ones with the characters T, F, R and the term DEF, where '**T**' stands for Top view, '**F**' stands for Front view, '**R**' stands for Right view and '**DEF**'. stands for Default view.
Such a layout is very handy to get a quick impression of what a new part looks like.

Note:
In a Layout with more than one view, one of the views will be the current (active) view. You can make a view current by clicking your left mouse-button inside the view: observe the blue line that appears around the active view. The thumb-wheels, view buttons and view commands apply only on the current view.

4.11 Items Visible



In this dialog you can select which items will be included in the View Window.
Four groups of items are present: General items, types of CAD-data, items about the complete Part and items that are different for each Operation. In front of each item you find a checkbox: checked means Visible, not checked means not visible.

Three **General items** are present:

The **Title** will be displayed on the screen and on the printed view, in the top left corner of the View. You can type a title in the edit box shown, if that is empty checking Title does not make a difference.

The **View Orientator** is the coordinate system icon (green cube with axes) displayed in the bottom left corner of each view. It helps you to understand from which direction you are looking at the geometry. This green cube does NOT indicate the zero point.

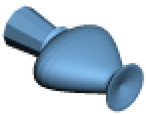
For machines with a 4th axis parallel to Y (option *Swap X and Y coordinates in the NC file* checked in the Advanced Machine settings) two View orientators are drawn.

The **Working Area** can be drawn, which is the working area of the machine selected for this part. Of course DeskProto does not know where you will set the workpiece zero point on the machine, so it will draw the part exactly in the middle of the working area. This will give a good indication how the part relates to the machine. The working area for your machine can be set in the Machine dialog.

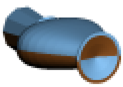
The second group shows all **CAD-data items**:

The **Vector curves** are all curves that are loaded as Vector data. The curves are drawn in gray, only when a curves has been selected in one or more vector operations it will be drawn in black.

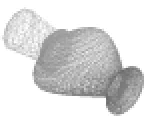
Vector curves in DeskProto are all polylines: the curve is built as a series of straight line segments. The sub-option **Show points** will indicate the end of each of these straight lines with a small point in the drawing. As you can see in the letter D (image on the left) no points are present on straight lines.



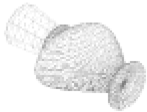
In a **Rendered geometry** drawing all triangles of the geometry definition are made "solid" with color. This offers a good understanding of your geometry, so it is the default way to draw all Geometry data.



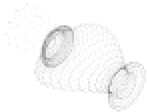
The sub-option **Show downward faces** will assign a different color to any triangles that are facing down (in other words: of which the normal has a negative Z-component). This option makes it very easy to check for Undercuts (areas where the cutter cannot reach), and to optimally rotate your geometry to reduce undercuts.



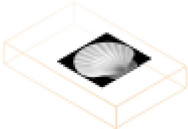
In a **Wireframe geometry** the triangles of the geometry are drawn with lines.
it is possible that you can't see the geometry very well because all the lines are confusing.



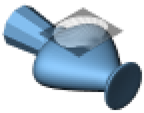
In such case the sub-option **Remove hidden lines** can be useful. This removes the lines of triangles that are hidden (obscured by other triangles).



In a **Points geometry** only the three vertices (corner points) of each triangle are drawn.



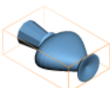
Bitmap concerns the Bitmap image that has been loaded. For large bitmaps DeskProto will display a simplified version (256x256 pixels) of the bitmap, in order to speed up drawing.



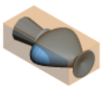
The sub-option **Translucent** draws the bitmap image in a translucent way, maintaining visibility of the items behind the bitmap that otherwise would remain hidden.

The **CAD zero-point** draws a gray orientator cube with three axes (as shown on the left) on the point where in the original CAD-data the zero point is located.

The group that includes the **Parts**-related items is the smallest:

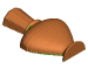


The **Material block** is drawn in orange lines. For standard three-axis machining this is a rectangular block, for rotation axis machining the block is a cylinder.



The sub-option **Translucent** draws the surfaces of the material block in a translucent color (the same orange/brown as used for the lines). This will show the block more clearly, maintaining visibility of the CAD-data inside.

The **Workpiece zero-point** orientator again is a small cube with three axes, drawn in dark blue and showing the position of the zero point on the machine. The position of the Workpiece zero point can be changed on the Zero point tab of the Part Parameters. Default position (as shown left) is on the front/left/top corner of the material block.

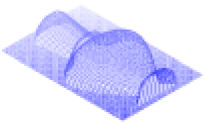


The **Simulation** checkbox allows you to add a Simulation of this Part, which will clearly show what the resulting model will look like. Checking this box will display the complete block of material and show the Simulated Operations dialog. Here all operations for this part are shown, and you can choose the operations to be processed in the simulation.

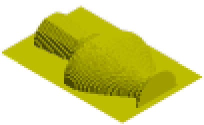
The last group shows the items that are different per **Operation**. So the selection of items in this group may be different per Operation. It is for instance possible to show the toolpaths of Operation Roughing and not show them for Operation Finishing, while both operations are visible.

In the list of **Operations** you can change the visibility of each Operation by clicking on the lamp icons (just as in the Project tree). Yellow (lamp on) means visible, gray (lamp off) means invisible. A red lamp indicates an error status for that Operation.
In the list you can also *Select* one or more Operations: making the line blue (meaning selected) by clicking on it. Two or more operations can be selected by keeping the Control or the Shift button depressed when clicking. The Operation Items that you will check and uncheck will apply ONLY to the selected operations.

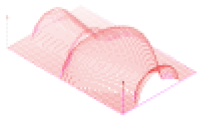
When you have selected two operations and some item is checked for one and not checked for the other, the checkbox will show a square instead of a V, like in the illustration above for the Toolpaths.



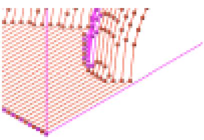
When you check the **Z-Grids** checkbox the Z-grids of the visible operations will be shown. The Z-grid is an intermediate representation of the geometry that DeskProto uses for it's toolpath calculations. The Z-grid will be drawn in lines.



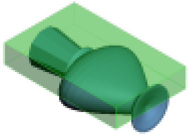
The same Z-Grid will be show for option **Rendered Z-Grids**. Now the Z-grid is drawn as a rendering, so you can clearly see that it is a 3D bar graph representation of the geometry.



When you check the **Toolpaths** checkbox the Toolpaths of the visible operations will be shown. The toolpath that is drawn is the same toolpath that will be sent to the machine: if there are any problems it should be possible to detect them now.



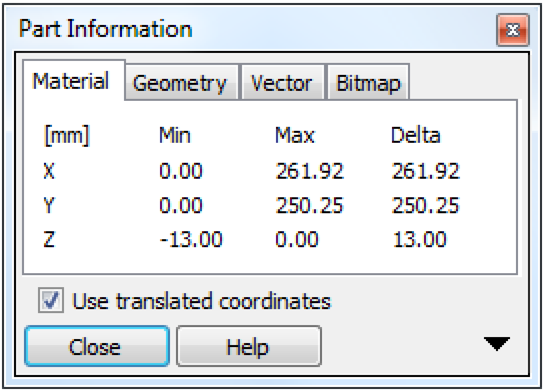
Toolpaths in DeskProto are all polylines: the path is built as a series of linear interpolations ("G1 movements"). The sub-option **Show points** will indicate the end of each of these straight movements with a small point in the drawing.



When checked the **Operation areas** will be shown: the rectangular bounding box of the operation in light green lines. The sub-option **Translucent** draws the sides of the area in a translucent green color. This will show the area block more clearly, maintaining visibility of the geometry inside.

The colors that are used for the various items in the example drawings above are the DeskProto default colors. These can be changed in the Colors tab of the DeskProto Preferences.
For all line drawings default fog may be applied to simulate depth, which can be set on that same location.

4.12 Part Information



Depending on which types of CAD data have been loaded in the project, one up to four tab pages are shown.
Each tab shows the Boundaries and Dimensions ("delta") , either for the complete part or for that type of CAD data.

All coordinate values and sizes are shown for the Part that results when the Part parameters have been applied (scaling, rotating, etc): these values are valid for the NC program file to be written and on your milling machine.
Only in case checkbox **Use translated coordinates** has been un-checked the final Part translation (as set on tab page Zero-point) will not be applied.

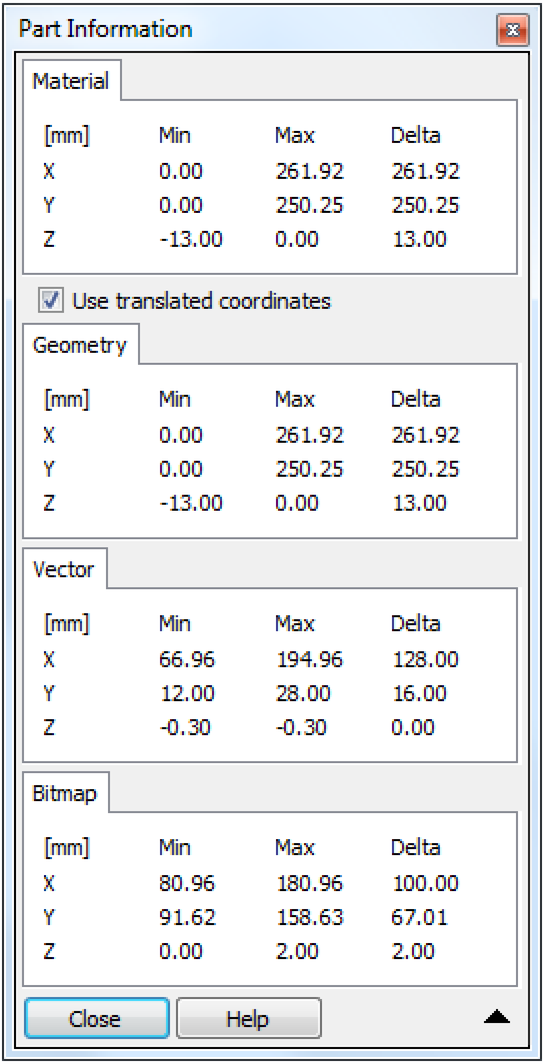
You can find the Min and Max values as present in the original CAD-data using button **File info** in the Project parameters.

Material:
The dimensions of the part as it will be machined. This is important information for you, as these are the dimensions of the Block of material that you have to prepare.
Vector:
The bounding box of all Vector curves that have been loaded. For the Z-values the machining depth is used that has been set in the Vector operations for this part. The deformation caused when projected on a 3D geometry is not shown.
Geometry:
The bounding box of all Geometry data that has been loaded.
Bitmap:

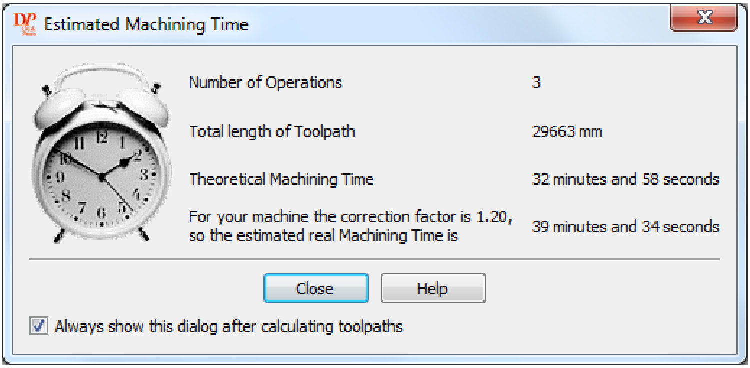
The bounding box of the Bitmap relief that will be created. The deformation caused when projected on a 3D geometry is not shown.

In all TAB pages coordinates are given for **Min**, **Max** and **Delta**. The min and max part values are handy when setting the workpiece zero point on the machine, the delta values (dimensions) can be used to prepare the block of material to be machined.
This dialog can stay open while you are working with the project.

The dialog can be viewed in two ways; as the small dialog box with tab pages (shown above), or as a large dialog box with all the information visible at the same time (shown below). You can switch easily between those two types of dialogs by pressing the button at the bottom on the right side (the small black arrow).


Part Information
Material
[mm] Min Max Delta
X 0.00 261.92 261.92
Y 0.00 250.25 250.25
Z -13.00 0.00 13.00
Use translated coordinates
Geometry
[mm] Min Max Delta
X 0.00 261.92 261.92
Y 0.00 250.25 250.25
Z -13.00 0.00 13.00
Vector
[mm] Min Max Delta
X 66.96 194.96 128.00
Y 12.00 28.00 16.00
Z -0.30 -0.30 0.00
Bitmap
[mm] Min Max Delta
X 80.96 180.96 100.00
Y 91.62 158.63 67.01
Z 0.00 2.00 2.00
Close
Help

4.13 Estimated Machine Time



With this option you get a rough estimation for the machining time that it will take to create your part. Be advised that this indeed is a very rough estimation, as the actual time will be influenced by many factors.

Theoretically speaking calculating the machining time is very easy: DeskProto knows both the length of the toolpath and the feedrate (machining speed), so length divided by feedrate results in a theoretical machining time. However, the real machining time is influenced by many factors:

- How long does the machine's controller calculate for one linear interpolation (calculating the separate speeds for each axis, a calculation that is needed for every movement) ?
- Does the machine keep up its speed (look-ahead buffer), or does it stop or slow-down in-between each two movements ?
- How fast can the machine accelerate and decelerate ? (especially important when stopping or slowing down after every movement).
- How fast is the data transfer from computer to machine ? (if you are using a 9600 baud serial line this factor will seriously slow down the process).
- Does the toolpath consist of small movements or long straight lines ? (in the first case it will in fact not even reach the desired feedrate as the distance is too short to accelerate to full speed).

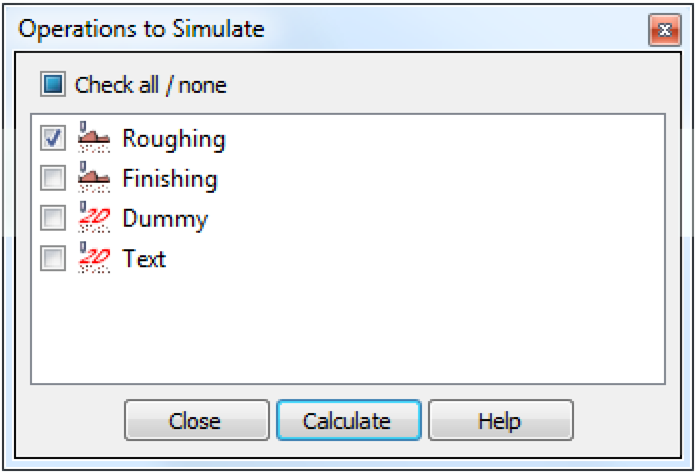
So the resulting real machining time cannot even be correctly predicted for one particular machine, as it will vary considerably depending on the characteristics of the toolpath.

DeskProto will convert the Theoretical machining time to an estimated Real Machining Time by multiplying by a machine dependent correction factor. As previously noted, this results in a rough estimation since the actual time is also dependent on the toolpath characteristics. The correction factor can be set at the Machine dialog (Library of Machines). In order to fine-tune you can time a few toolpaths and correct this factor accordingly.

This dialog will automatically pop up immediately when a toolpath has been calculated.
In case you do not like that: remove the checkbox before "Always show this dialog after calculating toolpaths".

You can make this dialog (re-)appear by selecting Estimate machining time in the Create menu.

4.14 Simulated Operations



This dialog is visible when a Simulation is visible on screen
Removing this dialog (red cross or button Close) will make the Simulation invisible.

You can open this simulation dialog by:
- pressing the Simulation button on the Toolbar (after which you can see that it is pressed)
- using the Calculate Simulation command in the Create menu.
- switching the simulation on in the Items visible dialog.
DeskProto will calculate a simulation in 3D, so you can rotate, pan and zoom it just like any other item on screen.

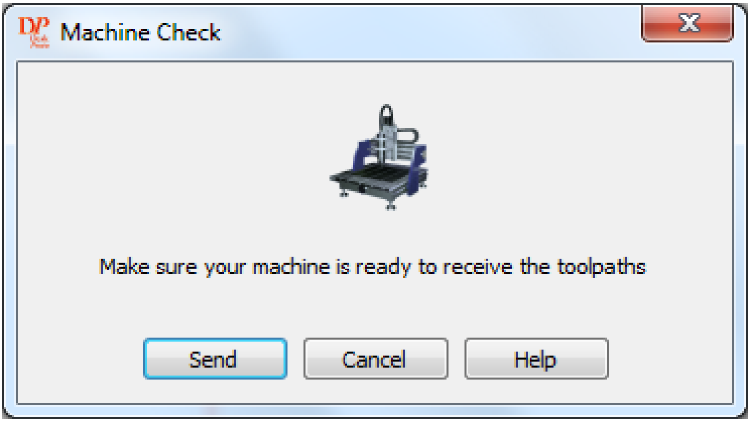
The simulation is made for the complete part. This Simulated Operations dialog will pop up in which you can select which operations should be included. When no operations are included the simulation shows the complete block of material. After checking an operation to be included you will need to start a calculation to see the result. Removing an operation from the selection causes the complete simulation to be reset.

Option **Check all / none** offers an easy way to include resp remove all operations in one click. When some operations have been selected and some have not, this checkbox shows a small blue square, like shown in the image above.

You can set the Simulation parameters in the Part parameters dialog.

Closing the Simulated Operations dialog, again pressing the Simulation button and switching the simulation off in the Items visible dialog will all make the simulation disappear from the view.

4.15 Machine Check

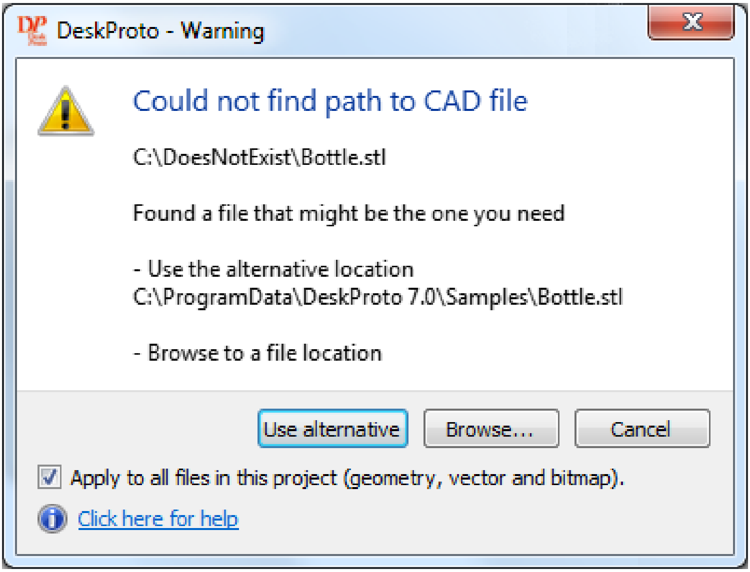


This dialog will be shown by command Send NC-program file to machine and Send current toolpaths to machine, both in menu Create > Extra.

It will be displayed to ask you if the machine is ready: material loaded, cutter loaded, zero point correctly set, machine ready to receive data.

After pressing Send DeskProto will start sending, and the machine may start cutting immediately.

4.16 File not found warning

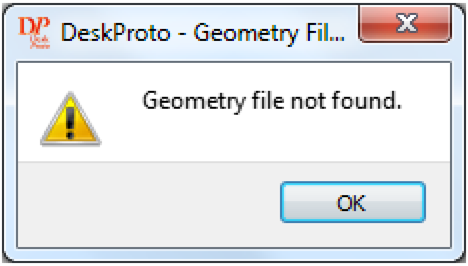


When opening a project file, the CAD file(s) that was (were) used will be opened again: the DPJ file does not store the data but only links to the Vector/Geometry/Bitmap files. These files must thus be found at the same place as where they were when the project file was saved.

If the CAD file cannot be found (for instance when the DPJ file was copied from a different computer system) then DeskProto will check if a CAD file with the correct name can be found in the current folder (same as the DPJ file) or in the default Data directory. If yes, DeskProto will ask you if it can use that file instead, using a dialog as shown above.
You then can choose to either **use** that file at the **alternative** location,
or **browse** to use a file on some other location.

The option **Apply to all files** is useful in case your project file contains more references to external files (Vector, Geometry and/or Bitmap). This option makes it possible to use the now location for all file references in the project.
If the option is not checked then this same dialog may be shown a number of times, once for every CAD file that could not be found.

If the file also cannot be found on any of these alternative locations, then this error-message will be displayed:

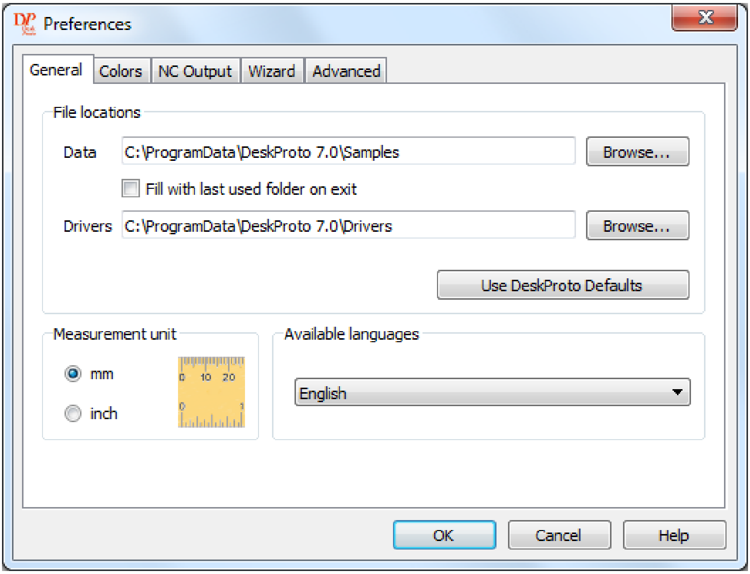


Which may of course also say "Vector file not found" or "Bitmap file not found".
After pressing the OK button the project will have been loaded without the missing CAD file(s).

4.17 Preferences

This is a dialog in which you can edit DeskProto's preferences. The dialog consists of 5 Tab pages.
These preferences are stored in the registry. Each user has its own set of preferences stored there.

General tab page



The **File-locations** listed here are the directories (folders) where DeskProto looks for the specified file types. These directories can be changed by typing the new directory's complete path-specification (i.e.: C:\DeskProto\ ) or by using the Browse button.

Data is the location where DeskProto will initiate the Load and Save dialog boxes for projects and for CAD data. The default setting for this option is the Documents (or My Documents) folder: a standard location in Windows. In this default location no sample files and projects are present: quickest way to find the sample files is via the DeskProto Start Screen (check "Use samples folder"). The screenshot above shows the Samples folder in the Data edit field.

While working with DeskProto this Data folder (the "**current directory**") may change: when loading from of saving to a different folder that folder will become the current directory. For instance after loading CAD-data from folder C:\Test, the default folder for loading more CAD-data or saving the project file will also have become C:\Test. This is very convenient, as it makes it easy to save all files for one particular project in one folder.

The Data directory comes with the extra option **Fill with last used folder on exit**, making DeskProto remember which folder you were working in the last time you used DeskProto.

Drivers is the location from which DeskProto will load all driver files (machines, postprocessors and cutters). Any valid new files that you copy to this folder will be automatically available after (re)starting DeskProto. Changing this location will make DeskProto unload all drivers (not delete them from disk though) and load drivers from the new location (if available). Because the open project uses drivers from that library, the open project will be closed before making this change. A new empty project will be created after the new library has been loaded or created at the new location.
Be careful though: DeskProto will not function correctly when it cannot find it's drivers.

The default setting for the folder Drivers is in the \ProgramData\ folder: a standard location in Windows meant for this type of files (folders in \Program Files\ can be accessed only by users with administrative privileges).
The folder is located in the root: *C:\ProgramData*
For some reason Microsoft has made it a hidden folder: to make it visible in File Explorer ('My Computer") select Organize >> Folder Options, tab page View and select the option "Show hidden files, folders and drives".

In Windows XP this folder is called: *C:\Documents and Settings\All Users\Application Data*
To make it visible in WinXP Explorer open Tools >> Folder Options >> tab View and select the option "Show hidden files and directories".

As this is a rather confusing situation it may be difficult to restore the DeskProto default after a change. So we have added the button **Use DeskProto defaults** to make resetting the file locations easy.

Using the option **Measurement unit** you can choose between metric (mm) and imperial (inches) for your DeskProto configuration. This setting will be applied in the user-interface and when loading and saving CAD-data: most CAD files do not state the units used and DeskProto has to assume that they are in the same unit as the one you define here.

Note: also check your milling machine’s documentation to see which units are needed for the machine, as that setting is independent from this preferences setting. Some machines expect NC files in mm, some will expect files to be in inches. Make sure to configure your postprocessor to use the correct units: this is completely independent of the choice that you make in these Preferences.
A few places that you can check in case of problems:
This Measurement unit in Options> Preferences> General tab, mm or inch.
The Distance unit in Options> Postproc> Movement tab, mm or inch.
The Feedrate units in Options> Postproc> Feedrate tab (many more options)
When your machine uses G-codes you can also check the Start commands of your Postprocessor:
If working in mm: use "G21 or G71 mm" to set the machine to use mm.
If working in inch: use "G20 or G70 inch" to set the machine to use inches.

Other **Available languages** than English are only present in case you have run a translated DeskProto Setup. This setup will create a Translations sub-folder in the DeskProto installation folder, with a sub-folder for that language, and it will copy translation files for a certain language to that folder. The name of the sub-folder is the two-letter codes for that language (conform ISO 639).

DeskProto currently supports the following languages:
de German
en English (this is the default, no Translations folder needed)
es Spanish
fr French
it Italian
ja Japanese
nl Dutch
ru Russian
zh Chinese

For each language "nn" at least the following file is **required**: *DeskProto_nn.qm*, containing the translated resources.

For instance “nl” for Dutch needs the file *DeskProto_nl.qm*
Complete file specification: *C:\Program Files\DeskProto 7.0\Translations\nl\DeskProto_nl.qm*

In addition the following translated files are **optional**:
- translations for the standard Windows buttons and dialogs, in file *qtbase_nl.qm*
- a translated Help file, in files *DeskProto_nl.qhc* and *DeskProto_nl.qch*
- translations for the custom wizards, in file *DeskProto_Wizards_nl.qm*

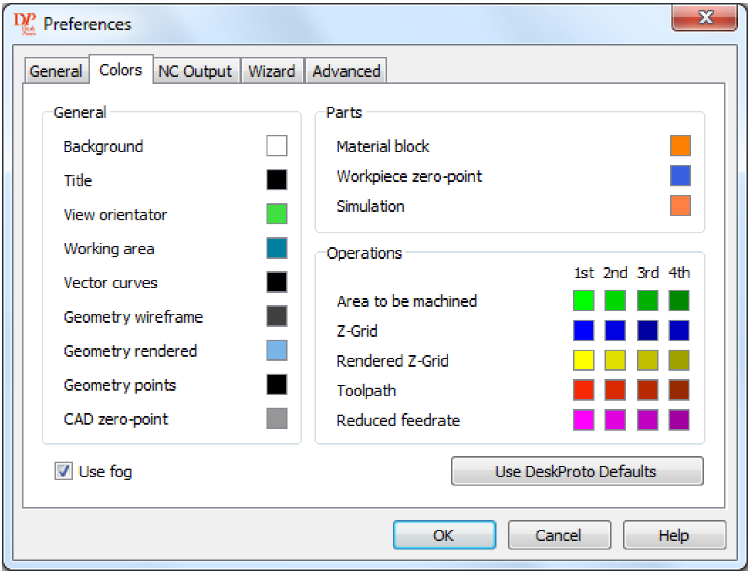
If such optional file is not present (or incomplete) the missing texts will be displayed in English.

The cutter files that are installed with DeskProto V7 already contain proper names (translations) for all these languages. This is different from the situation in previous DeskProto versions, where for each language a completely different series of cutter files was used.

The translated Setup will also copy translated versions of the license agreement and the Readme file:
License_nl.rtf and *Readme_nl.rtf*

Contact your local dealer to see if a translated version for your language has become available.

Colors tab page

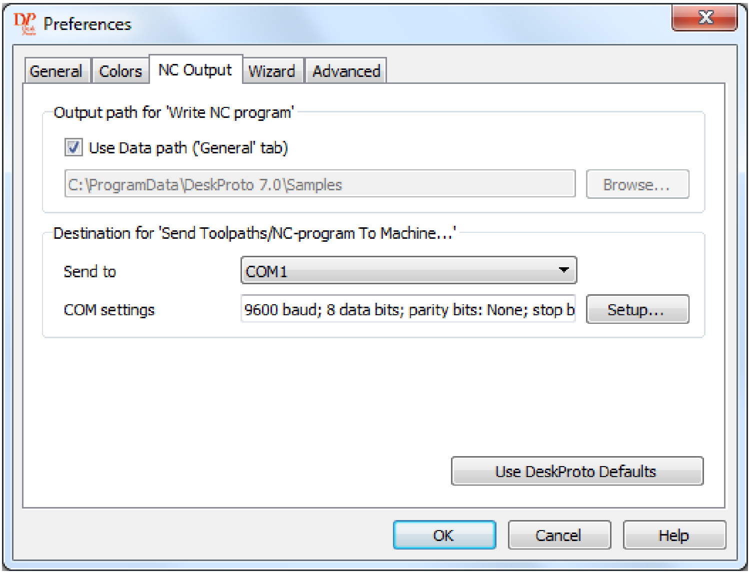


The tab page Colors makes it possible to customize all colors used on the DeskProto screen: all color fields are buttons that you can click on to open the Select Color dialog. As you can see different operations can be assigned different colors. In case of many operations: the colors for operation No. 1 will also be used for No. 5, 9, 13 etc.

Checking the option **Use fog** applies fog to all drawings using lines. This technique (also called 'depth cueing') improves the perception of depth on your two-dimensional screen by making distant lines more vague (as if obscured by a light fog).
On Black and White printers this may cause the lines to be printed like dotted lines, resulting in a blurry print. Therefore you might want to temporarily switch off the fog when printing. On computers with a video card which does not support OpenGL this option might not have effect.

In case you have made a giant mess of your color settings you can press **Use DeskProto defaults** to restore the default settings of DeskProto for all colors (these 'hard defaults' are stored internally in DeskProto and can not be changed).

NC Output tab page



These preferences are meant to configure the NC-output, so for exporting the toolpaths to NC files or other destinations.

With the first preference you can fine-tune the output path. Normally DeskProto exports the NC files to the same folder from where the project and/or the geometry file were opened (the 'current' directory). This is when the option **Use data path** is checked.
After unchecking this option you can type or **Browse** a fixed directory to be used for writing the NC files to. This is handy in case you want all NC files to be stored on the same place (for instance a folder on the PC next to the milling machine, or a USB stick that will be plugged into the machine).

The second preference refers to the commands Send NC-program File to Machine and Send Current Toolpaths to Machine in the Create menu. By default these two commands are disabled, here you can enable them by determining which process has to be started after giving the command.

Send to

This combo box shows four types of choices: Printer Driver, a Hardware Output port, an External program or None. Each option comes with it's own sub-settings:

1- In case you configure **None**, the commands Send NC-program To Machine and Send current toolpaths to Machine will not be available (grayed out), and the button "Send toolpaths to machine" is not present.
No sub-settings are available. This is the default situation.

2- Selecting a **Hardware Port** (either serial **COM..** or parallel **LPT..**) means that the NC-program will just be copied to that port after the command is given. This can be for example LPT1, or COM2. The button "Send toolpaths to machine" will show a machine icon (*see Note 1 below*), and the tooltip for that button will show the configured destination.
For a Centronics Printer port (LPT1 or LPT2) no settings are needed. In case of a COM-port you need to configure this port according to the specifications of the milling machine. Use the button **Setup** for the standard Windows **COM port settings** dialog (setting values like 'Baud Rate', 'Data Bits', 'Parity', 'Stop Bits' & 'Flow Control').

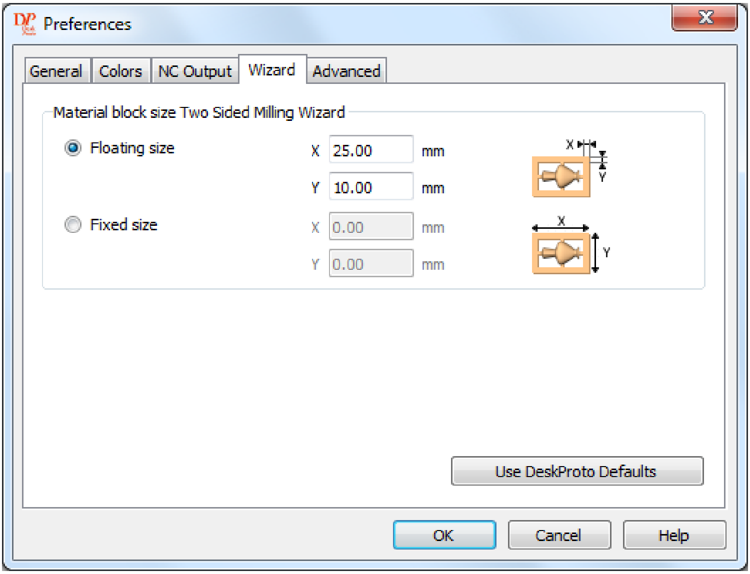
3- Selecting **Printer Driver** can be used for machines that are accessed via their own printer drivers (for instance most Roland machines). This also works for USB machines that cannot be accessed via a Port. The button "Send toolpaths to machine" will show a machine icon (*see Note 1 below*), and the tooltip for that button will show the configured destination.
Here you have to select the correct **Machine name** from the list of available printers (or rather of all printer drivers that have been installed on your PC). Take care not to select a 'normal' printer as this may result in hundreds of pages to be printed on paper.
After sending the NC-file to the machine this way, the job status can be followed using the standard Windows Printer Properties tools. On some PC's it may be needed to select "Print directly to printer" in the Advanced Printer properties of the selected printer driver.

4- After selecting the option **External program**, you can use the **Browse** button to define which program has to be used. The button "Send toolpaths to machine" will show the desktop icon of that program (*see Note 1 below*), and the tooltip for that button will show the configured destination.
Here you can browse to any program file (EXE of COM) on your computer. After the command Send NC Program to machine, DeskProto will start this program, with the name of the NC-program file to be used as command line parameter.

The idea is to configure the control software of your CNC milling machine here. This works for many programs, for instance:
- Kay (configure "*C:\Galaad\KAY.EXE*", or "*C:\Program Files (x86)\Galaad\KAY.EXE*").
- PlanetCNC (configure "*C:\Program Files\PlanetCNC\PlanetCNC64.exe*", or PlanetCNC32.exe on 32bit windows).
- WinPC-NC (version 3.01 or newer, configure "*C:\WinPCNC_USB\WinPCNC.EXE*").
However, it does not work for all control software, for instance not for Mach3: the control software must support the use of a command line parameter with the name of an NC file, and Mach3 does not.
In fact you can select any program, for instance a milling simulation software, or (for the diehards) a plain text editor like Notepad, allowing you to change the NC-program that DeskProto has just created.

Note 1:
It is possible to configure a **custom icon** to be shown on button "Send toolpaths to machine" . That is ideal for machine builders that give (or sell) a DeskProto license with their machine: they can make DeskProto show their very own logo !! Configuration is easy: copy an image file called *machine.png* to the folder that also contains DeskProto.exe (normally *C:\Program Files\DeskProto 7.0*) and that image will be shown on the button. Scaled down to the proper size (default 32x32 pixels).

Wizard tab page

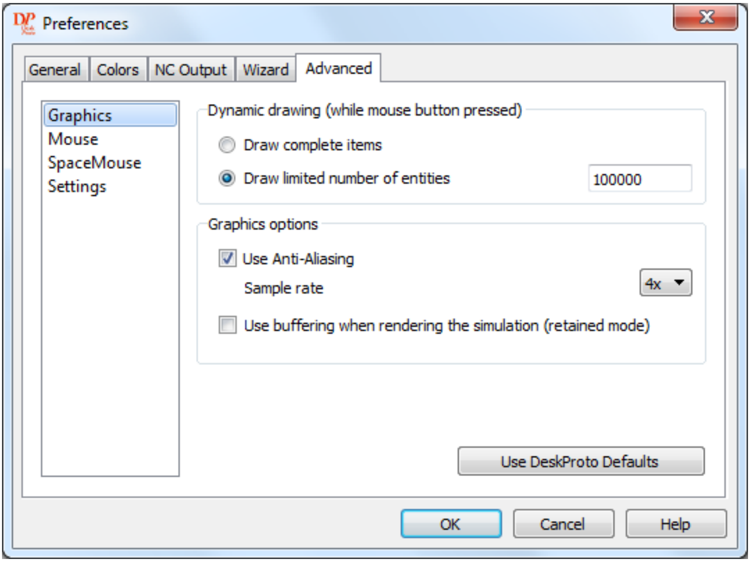


The tab page Wizard contains a preference that you can set for the Two-Sided Milling Wizard: the size of the material block. This tab page is not available in the Free edition and the Entry Edition. Two options are available:

- **Floating size**. The block size needed of course depends on the size of the part that you want to create. DeskProto offers a flexible solution here, called Floating size: around the part (including the support tabs) a frame is added of a certain width. In the edit boxes for X and Y you can enter which frame size to use. This is the default choice for this option.
- **Fixed size** is useful in case all your parts have almost the same size: you then can use standard blocks of a certain size, to be defined in the X and Y edit box.

Only the X and Y block size can be set here, the Z dimension is set in the wizard.

Advanced tab page



This tab page of the Preferences contains three or four different sub-pages, which you can select by making it active (blue background) in the list on the left. Three sub-pages are always shown: **Graphics**, **Mouse** and **Settings**; page **SpaceMouse** is available only when DeskProto finds a SpaceMouse attached to the PC. Help for all four sub-pages is given below.

First the **Graphics** preferences will be explained, shown in the picture above.

Dynamic drawing

Dynamic drawing is the drawing that is done while the left mouse button is pressed inside the view window or while using the thumb-wheels: you see the image change while moving the cursor. Choosing the option **Draw limited number of entities** will decrease the number of entities that is dynamically drawn, and so will increase the speed with which you can move the geometry and other items (the term entities used here stands for points, lines and facets). If several Items are drawn, the maximum number that you enter is split up over all visible items. In case you set the number of entities to zero only the bounding box of the geometry will be displayed during dynamic drawing.

The optimum number of entities depends on the capabilities of your graphics card: an OpenGL card featuring hardware rotation etc. can handle a large number of entities at high speed. For a simple graphics card the number must be set lower to achieve real time results.

Graphics options are in fact OpenGL options:

Drawings on the screen are displayed using pixels, for instance a line is drawn as a series of black pixels on a white screen. As the pixels on your monitor are located in a rectangular grid, a curved line will be displayed as a staircase. **Anti-Aliasing** is a technique to make this staircase effect less visible by adding pixels with an intermediate color value: in the example just mentioned this will be in various grades of gray. This shade of gray to be used for a pixel is calculated using multi-sampling: each pixel is divided into multiple sub-pixels, for each sub-pixel a gray value is calculated, and the average resulting gray value is used for the pixel to be displayed.

The more samples are used, the better the resulting quality, however the more calculation time is needed. You can set this **Sample rate** as a detail setting: the combo box will show only the rates supported by your graphics card. We advise a rate of four: higher rates will cost much time where the difference will be marginal.
In case your graphics card does not support anti-aliasing at all, this option will have been grayed out.

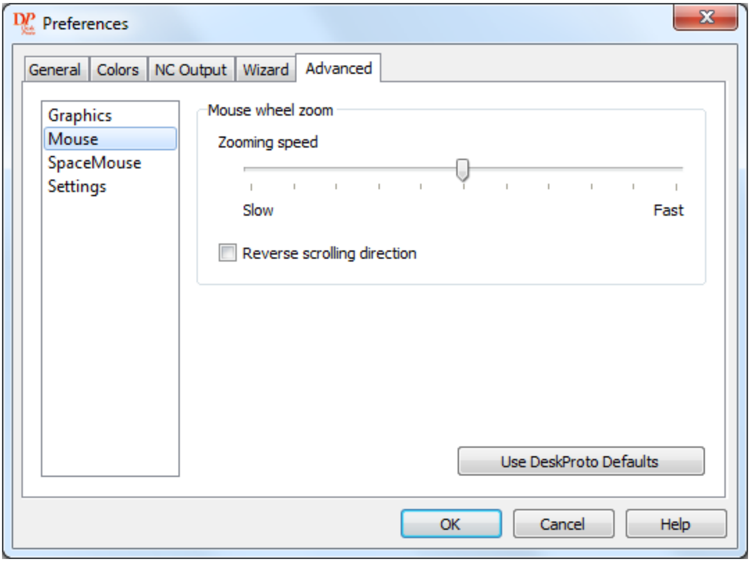
The option **Use buffering** is meant to optimize the drawing speed for your typical **simulations**.
A simulation in DeskProto is calculated and drawn as a large set of small triangles (defining it's outer surface). This is the same type of entity as used to draw the geometry. Only the number of triangles in most cases is higher for a simulation, making the drawing quite slow.

Normally (so without buffering) color values for these triangles are generated for the current view settings and are sent to the card as 2D data. The graphics card just displays the resulting 2D image, and does not keep 3D data in it's video memory. This will of course cost time: for each new view all triangles have to again be generated and sent over to the graphics card.

Using Buffering means that all triangles are stored in RAM as a sort of "geometry model", and for drawing are sent to the 3D **video memory** of the graphics card as 3D data (a 'buffer'). Changing the view then can be completely handled by the graphics card: no need to again send all data from the application. Which is of course MUCH quicker.

The drawback however is that much more memory is needed for this buffered method. When more memory is needed than available (either RAM or video memory), Windows will need to swap to hard disk. And that will make this nice new method even slower than the un-buffered method. So buffering is advised only in case you have small simulations: where "small" needs to be considered relative to the amount of RAM memory in your PC and video memory on your graphics card.

One more graphics option is available: the **line width** (default 1 pixel) can be changed for all lines in the drawing (curves, blocks, toolpaths). This has been implemented a "hidden feature", using a registry entry.
It can be switched on by editing a value in registry key HKCU\Software\Delft Spline Systems\DeskProto\7.0\Preferences\Advanced\
Registry value DrawingLineWidth may have a value from 1 (default) to 99.
Changing this value needs to be done before DeskProto is started.
Only edit the registry when you are qualified and know exactly what you are doing !

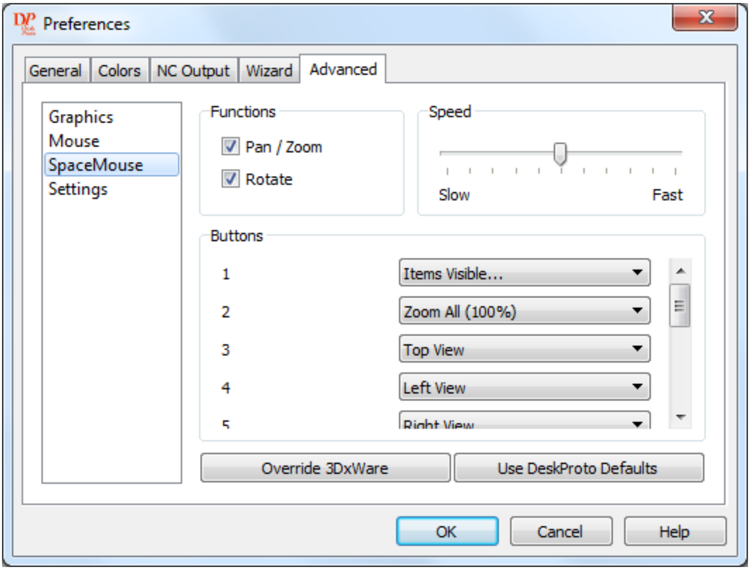


The picture above shows the **Mouse** sub-page of the Advanced Preferences.

The only Mouse setting that you can configure is the **Mouse wheel zoom**: rotating the mouse-wheel (scrolling) when the cursor is in the View window will make DeskProto zoom in or zoom out.

The **Zooming speed** sets the zoom percentage for each rotation step of the mouse's scroll-wheel.

Scrolling from back to front will zoom in: your finger movement pulls the part on screen closer. Scrolling from front to back will zoom out: your finger movement pushes the part away.
The option **Reverse scrolling direction** reverses this situation.



The picture above shows the **SpaceMouse** sub-page of the Advanced Preferences.
It will be visible **only** in case a SpaceMouse is connected to the PC.

These settings are meant for the 3Dconnexion SpaceMouse series of products (formerly called Logitech SpaceMouse, now offered by Logitech daughter company 3Dconnexion). The sub-page will be present only when on program start DeskProto detects that such SpaceMouse is present on your computer. The product is also called 3D Mouse and SpaceNavigator.

The image above shows the most basic version. The device stands on your desk, the handle (rubber knob) on top is pressure-sensitive in six directions: front-to-back, left-to-right, up-and-down, pitch rotate, roll rotate, yaw rotate. Applying pressure in one of more of these directions will make the 3D part on screen move and rotate in these directions. Which is a great help for CAD designers.

This use of a SpaceMouse in fact is so straightforward and intuitive that no settings seem to be required. Still DeskProto offers a few.

Functions:
Of the six degrees of freedom just described three are linear movements and three are rotations.
Un-checking **Pan/Zoom** disables the linear movement commands
Un-checking **Rotate** disables the rotation commands.
When you un-check both only the buttons still work - which does not seem to make sense though.

Speed:
Just as for the scroll-wheel on the normal mouse you can set how fast the movement of the part should be, here in response to the pressure applied to the mouse's handle.

Buttons:
Each SpaceMouse features (at least) two buttons: for the basic model button 1 is the **Left button** and button 2 the **Right button**. For each button you can select the action that needs to be done when it is pressed. Each SpaceMouse model has a different number of buttons: of course you can't use more than your SpaceMouse offers.

DeskProto sets defaults actions for twelve buttons:

1. Menu -> Items visible dialog
2. Fit -> Zoom all
3. Top -> Top view
4. Left -> Left view
5. Right -> Right view
6. Front -> Front view
7. Bottom -> Bottom view
8. Back -> Previous view
9. Roll cw -> Rotate image 90 degrees clockwise
10. Roll ccw -> Rotate image 90 degrees counter-clockwise
11. Iso 1 -> Isometric view
12. Iso 2 -> Default view

You can change any of these button definitions by selecting a different action from the pull-down list for that button. After changing one or more button definitions you can use button "Use DeskProto Defaults" to reset to the 12 actions shown above.

In case you are not sure **which button corresponds to which number** you can simply press that button with this Preferences page opened: in the dialog shown above DeskProto will then make the number for that button red. In the screenshot above you can see that the "2" is displayed in red.

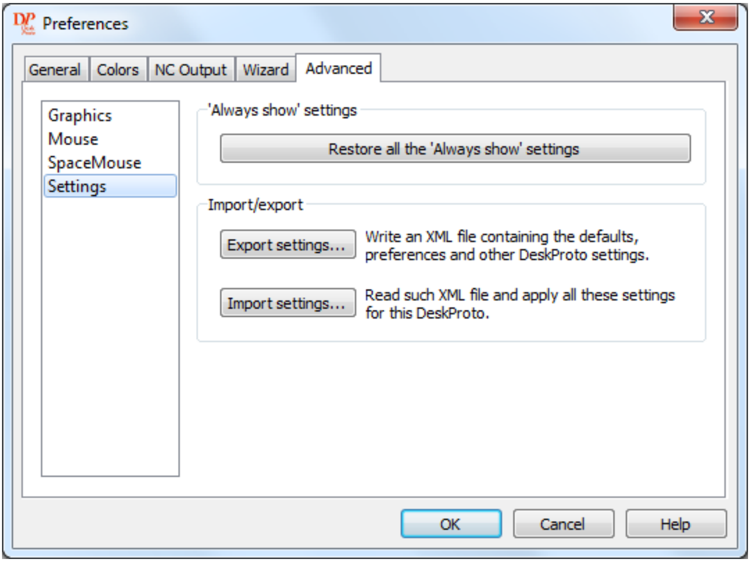
To understand the function of button **Override 3DxWare,** it is important to know is that DeskProto does **not** use 3Dconnextion's software development kit (**3DxWare**), and also does **not** use the 3DxWare driver that 3Dconnexion ships with this device. Reason is that this software is not compatible with the QT toolkit that DeskProto also uses.
Disabling the official 3DxWare driver (only for application DeskProto.exe) is achieved by copying file
C:\Program Files\DeskProto 7.0\SpaceMouse\Win.xml to
C:\Users\Name\AppData\Roaming\3Dconnexion\3DxWare\Cfg\DeskProto.xml
(where Name is the name of the user).
This copy action is done automatically, by the DeskProto Setup.

However, in the 3DxWare driver (3Dconnection Properties) a user can create a configuration for application DeskProto, which will over-write the one that was copied during Setup. When that happens both drivers will be active at the same time, resulting in zooming activity during all rotation commands: *the view will rotate and zoom at the same time, while it only should rotate.*

Button **Override 3DxWare** makes it possible to fix this problem: it will again copy the DeskProto version of the XML file, as described above. This disabling of the driver will only be effective for DeskProto.

This fix does not work correctly for old 3DxWare drivers (it did not work with a V4 driver), so if it does not work for you then you may need to upgrade your driver (it worked OK with a V10 driver).

Button **Use DeskProto Defaults** allows you to reset all setting on this page to the original DeskProto defaults.



The picture above shows the **Settings** sub-page of the Advanced Preferences.

In DeskProto many warning messages contain a checkbox called "Always show this message".
These are optional warnings: when you remove the mark from the checkbox and press OK, the warning will no longer be issued each time the same situation occurs. However what to do when you want the warning restored ?
Using button **Restore the 'always show' settings** you can restore all warnings (so it will again set the mark in all these checkboxes).

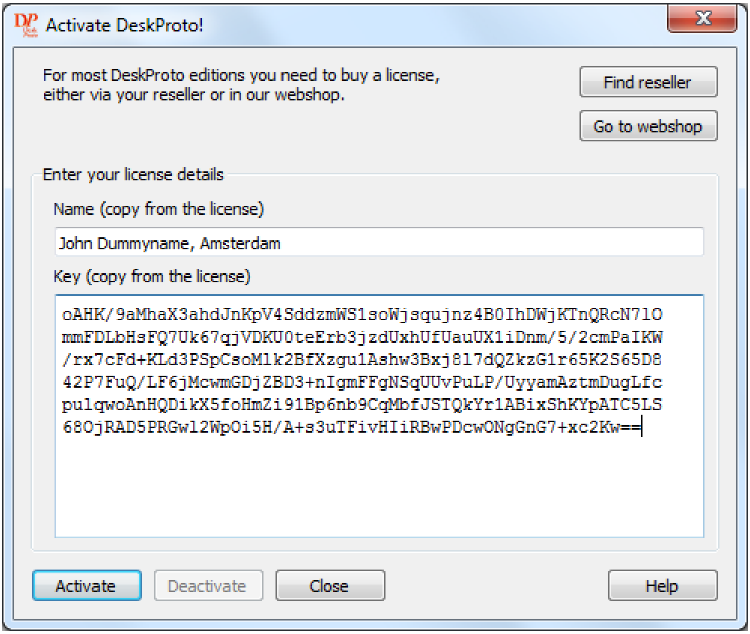
All DeskProto defaults and preferences are stored in the Registry, conform the specifications for Windows applications as made by Microsoft. This makes it difficult to extract these settings, for instance for backup purposes, or in order to give DeskProto exactly the same workspace on a number of PC's.

This is where the options **Export settings** and **Import settings** come in: these tool allow you to export DeskProto's Registry settings to a file, and also to import all these settings from a file that has been created previously or on a different PC.
The file format used is an XML file, default filename is *deskproto_export.xml*

The use of these commands is straightforward: button Export settings will export an xml file, button Import settings will import one.
The export option will write the current settings, so for the current version of DeskProto.
The import option may accept files written by older DeskProto versions, depending on how much older.

Note: the File locations (Data location, Drivers location, NC-File location) will not be included in this Settings file. Reason is that these will be different per user: like
C:\Documents and Settings\Jones\My Documents\
and
C:\Documents and Settings\Smith\My Documents\

4.18 Activate license



The **Free** Edition of DeskProto may be used without buying a license. Users who need more functionality can buy a license to activate (unlock) one of the higher Editions: **Entry**, **Expert** or **Multi-Axis**. The Free edition allows you to evaluate (trial) these higher Editions, though when trialling watermark (the "Trial cross") will be visible in all machined parts.

In all cases you need to first download the DeskProto Setup file from our website www.deskproto.com and instal DeskProto on your computer. This Setup file is the same for all Editions of DeskProto.

A **free DeskProto license** is given to any user: you are welcome to use the Free Edition of DeskProto free of charge.

A **paid DeskProto license** can be bought either via a reseller or via our webshop. After buying you will receive the license (a PDF file) containing two strings: a Name and a Key.

The **Name** contains the name of the buyer (either a person or a company) and his/her/its location (city, village): information that will be clearly shown at each program start.
The **Key** is a code of 340 characters, containing information all license information. Each Key is valid **only** for the Name on that same license.

The Activate dialog is meant to activate a DeskProto license and unlock the extra functionality for the Edition that you have bought. Both the complete **Name**-string and the 340 character **Key** have to be entered ***exactly as given***, including case (upper or lower), spaces, commas, points, etc. Any small difference will make DeskProto refuse to activate. So use Copy/Paste to enter this information in this dialog.
After filling both fields you can use the **Activate** button to make DeskProto activate the license.

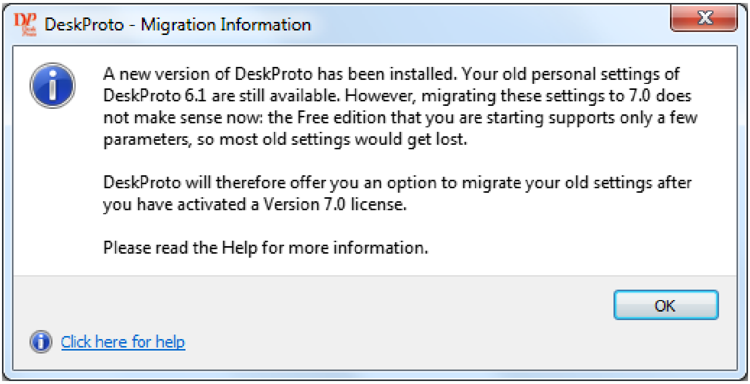
Make sure to carefully **save and backup** the license: you will again need Name and Key when (for instance) you buy a new computer.

A registered version of DeskProto will clearly show the Name of it's owner: in case that is not you then you are running an illegal copy!

After Activating it is also possible to **Deactivate**, allowing you to again convert the program from paid license to free license.
You my Activate and Deactivate as many times as you like: information about the previous licenses will simply be deleted.
Activating a new license is also possible without Deactivation first.

Both for Activating and for Deactivating **administrative privileges** are required.

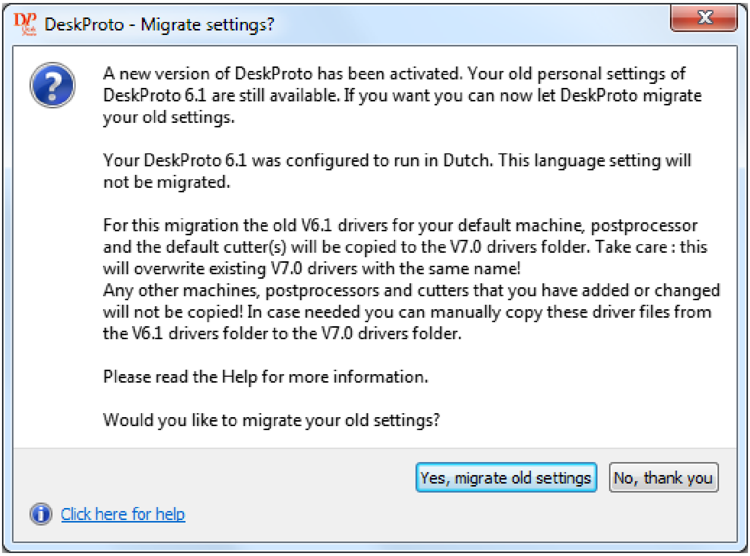
4.19 Migrate Settings



DeskProto stores many of the user's defaults and preferences in the Registry, conform Microsoft's guidelines for Windows programs. When you install a newer DeskProto you can choose whether or not you want to keep using the settings of your old DeskProto, so whether or not to migrate them to the new version.

Two different situations are possible:
1. You have installed **a new version** of DeskProto, next to an older version number. For instance you have installed a new DeskProto V 7.0 on a PC where a DeskProto V 6.1 was already present.
Then DeskProto will pop up the first **Migrate Settings** dialog as shown above.
2. You have reinstalled **the same version** (for instance you have installed a new DeskProto V 7.0 over an existing DeskProto V 7.0). Then the Keep Settings dialog will be shown.

As explained in the dialog above, at this point DeskProto cannot yet migrate the settings: you have now started the Free edition that only supports a limited number of parameters. Migrating can be done after you **Activated** your V7 license and thus again are using the correct edition.



When you activate your DeskProto V7 license the second **Migrate Settings** dialog (shown above) will pop up, allowing you to migrate the old settings. This dialog will be shown only the first time that you activate V7.

The personal settings for each DeskProto version are stored on a different location in the **Registry**, and you have the option to copy your old settings (in the above screenshot V 6.1, however for you it may be a different version) and keep using them for DeskProto V 7.0
This concerns all settings that you made in the Preferences and in the default Project, default Part and default Operation(s).

When you answer **Migrate old settings**, DeskProto will read these settings of the old DeskProto and save them as settings for the new version. This will be done as far as possible: not all settings are compatible between different versions.
In order to let DeskProto V7 use the old defaults a few **driver files** will be copied as well: for the default machine and postprocessor, and for the default cutter(s). Be warned: in case already present in the V7 drivers folder, this will replace V7 driver files with the same name! The V7 driver will be renamed to name.bak1 (or name.bak2, etc)
Other machines, postprocessors and cutters that you (may) have added are not migrated.
The Language setting will not be migrated as it might not be available in the new DeskProto. If needed you can manually change that in the Preferences.

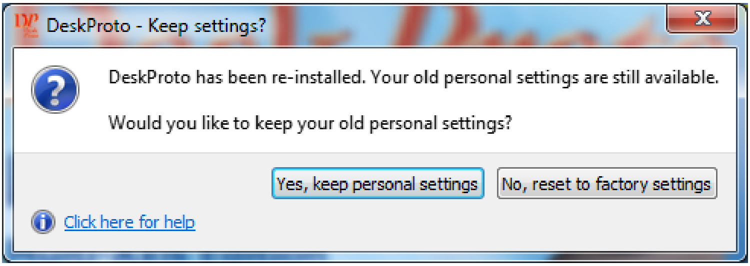
When you answer **No thank you**, nothing will happen, and the new DeskProto will start with system default values.

Manually migrating is possible as well:

In DeskProto every Machine, Postprocessor and Cutter has it's own **Driver file**: name.mch for a machine, name.ppr for a postprocessor and name.ctr for a cutter. These files are stored in the DeskProto Drivers directory.
You can simply copy these files from the old Drivers folder to the V7 Drivers folder. Do not copy all files, as that may remove improvements were made in DeskProto V7. When you sort the old drivers on date it is easy to see which drivers you have added and/or changed.

Migrating the settings (preferences and defaults) is possible via the Preferences, tab Advanced. Under Settings you can find buttons **Export settings...** and **Import settings....** In the old DeskProto you can export all settings to a file (XML file), and in the new DeskProto you can import that file.

4.20 Keep settings



DeskProto stores many of the user's defaults and preferences in the Registry, conform Microsoft's guidelines for Windows programs. When you again install DeskProto you can choose whether or not you want to keep using these settings.

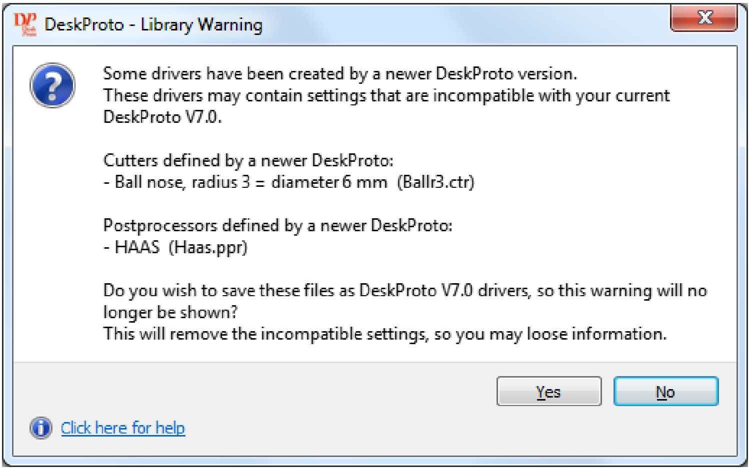
Two different situations are possible:
1. You have reinstalled **the same version** (for instance you have installed a new DeskProto V 7.0 over an existing DeskProto V 7.0). Then this **Keep Settings** dialog will be shown, further see below.
2. You have reinstalled **a newer version** of DeskProto next to an old one. For instance you have installed a new DeskProto V 7.0 on a PC where a DeskProto V 6.1 was already present.
Then DeskProto the Migrate Settings dialog will be shown.

This Keep Setting dialog offers you two options:

- when you answer **Keep settings**, nothing will happen:
DeskProto will keep using the settings as they were.

- when you answer **Reset settings**, DeskProto will delete the old settings and replace them by the DeskProto system defaults.
So when you have made a giant mess of your settings this is an easy way to delete them all.

4.21 Newer Drivers warning



This warning is given at program start, when DeskProto has detected that one of the Driver files (each containing a definition for a Cutter, a Postprocessor or a Machine) has been created by a newer version of DeskProto.
For instance: your DeskProto V7.0 opens a cutter file that was created by a DeskProto V7.1

This newer version of DeskProto may of course offer new features (not yet known at the time when this Help information is written).
And so this (these) driver file(s) may contain settings that you current DeskProto will not understand. In most cases this is no problem: your current DeskProto will simply ignore a feature that it does not recognize.

Take care though: new functionality may also be added to an existing feature. For instance: DeskProto V7.1 will support "placeholders", allowing you to add information like program number, part size, date, etc to the NC file. These placeholders (in case present) will be read by your current DeskProto V7.0, and will make the resulting NC file useless. So when this warning is given for the postprocessor that you use: check the resulting NC file for correctness.

This warning will be shown each time that you start DeskProto, unless you select Yes:

in that case these drivers files will have saved again, now as DeskProto V7.0 files.

For the toolpaths and NC files created with your current DeskProto V7.0 answering Yes or No won't make any difference.
However, after answering Yes the extra settings will no longer be available when the drivers files later would be copied to the Drivers folder of newer DeskProto version and read by that newer version.

V Concepts

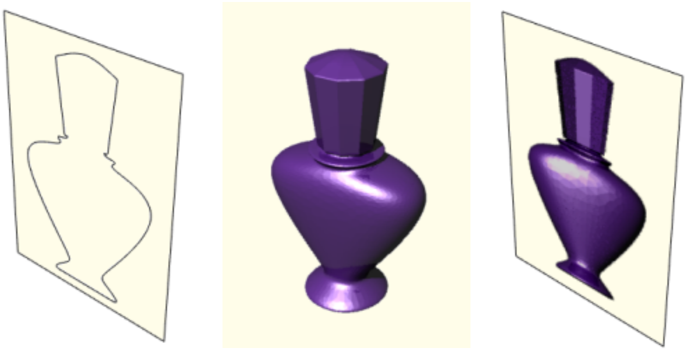
5.1 CAD Data

Vector data **Geometry data** **Bitmap data**

DeskProto offers **CAM** (**C**omputer **A**ided **M**anufacturing): the ability to calculate CNC toolpaths for a part that you have designed. The first step is this process in all cases is loading some CAD-data, as DeskProto does **not** offer **CAD** (**C**omputer **A**ided **D**esign) functionality. Three different types of **CAD-data** are supported:

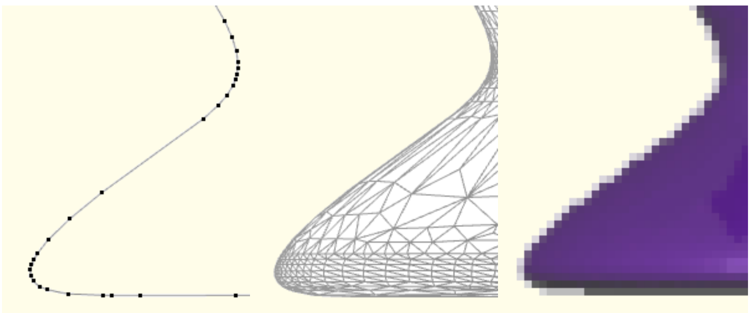
- Vector data One or more curves: points in space connected with straight line segments (or arcs).
- Geometry data A collection of small triangles that describe the outer surface of a 3D shape.
- Bitmap data A grid of colored pixels on a flat plane, forming a 2D image.

The images above for Geometry data and Bitmap data are identical, because on a 2D screen or page we can only show bitmap images. Bitmap data for DeskProto normally will be a digital photo, a company logo or similar. We used this image to show the difference between 2D data and 3D data:



Vector data **Geometry data** **Bitmap data**

When looking at the CAD-data from a different viewpoint the difference between the Geometry data (3D) and the Bitmap data (2D is clear: a real 3D bottle versus a flat drawing of the bottle on paper.
Vector data for DeskProto in most cases will also be a 2D drawing, however DeskProto will also accept 3D curves as Vector data.



Vector data **Geometry data** **Bitmap data**

The images above clear show the difference between these three types of CAD-data.
Both for Vector data (points connected by straight lines) and Geometry data (collection of triangle surfaces) you can zoom in without loosing detail. For Bitmap data this is different: when zooming in the pixels will became large squares, and what seemed a smooth curve becomes a staircase.

For each type of CAD data DeskProto offers a (slightly) different work-flow:

- a Vector project uses Vector operations, and offers Vector settings in the Part parameters
- a Geometry project uses Geometry operations, and offers Geometry settings in the Part parameters
- a Bitmap project uses Bitmap operations, and offers Bitmap settings in the Part parameters

The wizard interface also offers different wizards for each type of CAD-data.

A very powerful feature in DeskProto is that it can combine these types of CAD data in one project:
a Geometry with Vector curves and /or with a bitmap. It is even possible to project the Vector curves and the bitmap onto the 3D geometry.
In the illustration above the **Coat-Of-Arms relief** Geometry data (*a free download from the DeskProto website*) was combined with 2D Vector data for the word "DeskProto" and an image of the clock as Bitmap data. You can of course use your own data for the text on the ribbon and the image on the shield.

5.2 Vector Data

DeskProto can load three types of **CAD data**: **Vector data**, Geometry data and Bitmap data.

Vector data consists of curves: points in space that are connected with straight line segments. In most cases a 2D line-drawing. *DeskProto will use the curves in the drawing to generate the toolpaths*. Ideal for 2D jobs, like milling 2D shapes out of sheet material, or engraving on a flat surface.
In order to create Vector toolpaths you need to use a Vector Operation in your project.

How to load a Vector file

Vector data files can be loaded in several ways:
- File menu command Load Vector file
- Toolbar button Load Vector file
- Button Add... on the Vector tab of the Project parameters
- The Wizard Basic Vector machining.

It is possible to load more than one Vector file, using the same command: after you have loaded a vector file the Load Vector file command will change to Add Vector file.

How to save Vector data

Vector data can be exported in two ways (for export only DXF is supported):
- File menu command Save Vector data as...
- Button Save as... on the Vector tab of the Project parameters

This will save all Vector data to one file (also when various Vector files had been loaded).
When saving the Vector data you may select which of the Part transformations you want to apply for the exported file. You can do so in the Save Vector Options dialog.

Vector File types

For Vector data DeskProto supports three file types:
- **DXF** the most widely used exchange format for **engineering**
- **AI** the most widely used format for **graphics design**
- **EPS** can be seen as an older version of EPS

For all three filetypes DeskProto supports (so can read) only a small subset of all entities that can be present in the file.

DXF

The acronym stands for AutoCAD **D**rawing e**X**change **F**ile.
It can contain many different entities, both 2D and 3D, DeskProto supports a small subset only.

Supported DXF entities for Vector data in DeskProto:
POINT
A point in the DXF file can only be selected for a drilling operation on that location.
LINE
A line contains a begin- and end-point, and will result in a tool movement from begin to end (linear interpolation).
POLYLINE and **LW POLYLINE**
A polyline is in fact a series of lines connected to each other. This is more efficient (endpoint line 1 = startpoint line 2) and gives more control over the toolpath sequence. A polyline may contain arc segments as well.
ARC
An arc entity contains the centerpoint coordinates, the radius of the arc, and angles for start and stop. Currently DeskProto will convert the arc to a polyline (many small line segments), as the toolpath part (postprocessor) does not yet support arcs.
CIRCLE and **ELLIPSE**
Same as the Arc: stored in DXF as Centerpoint(s) and radius, converted by DeskProto to polylines.
SPLINE
A spline curve (Bezier, B-spine or NURBS). DeskProto will convert the spline to a polyline when importing.

DeskProto will also import CAD-layer information (if present). When selecting curves you can make each layer visible or invisible.

Text that is stored as a series of ASCII character with a font definition cannot be read by DeskProto.

As a nice extra feature DeskProto also supports **3D Vector files**. The entities mentioned above normally do not contain any Z-coordinates: you will set the Z-values in the Vector Operation parameters. A 3D Polyline however does contain Z-values, and DeskProto can import these. The

toolpath then will follow this 3D Polyline, and the Z-values as entered in the Vector Operation parameters will be taken relatively to the Z as defined in the file.
When you open a DXF file containing Z-coordinates then DeskProto will report that is has detected Z-values in the file, and will ask you if you want to use them. After Yes the Vector operation will produce a 3D toolpath, after No the Z-values will be ignored. This setting "**Use Z-values**" can also be found on the Vector tab of the Project parameters.
This option can also be checked for vector files that do not contain Z-values: DeskProto then will use Z=0 for all points on the vector curves. This will influence the result when both vector files and geometry files are loaded as the CAD-data zero level of the geometry file is used, which in most cases will not be the top of the block.

Note that DeskProto also supports Geometry data in DXF files.

AI and EPS (Postscript)

These acronyms stand for **E**ncapsulated **P**ost**S**cript (a file exchange format) and **A**dobe **I**llustrator (native format of the most widely used graphics design program). Both are in fact variations on the **PostScript** format, and for both files DeskProto supports the same subset of entities.
Since Adobe Illustrator Version 9 a completely different format for AI has been used. So AI files need to be saved/exported for **AI8** to make them readable for DeskProto. Loading an AI file in the new format will result in the error "The version of this file type is not supported".

Supported Postscript entities for Vector data in DeskProto:

POINT

A point in the Postscript file will result in a drilling operation on that location.

LINETO

A line in Postscript contains only an end-point, and will result in a tool movement from the current position to that end (linear interpolation).

CURVETO, CURVETO_USESTART, CURVETO_USEEND

Also a tool movement to the defined end-point, however now a Bezier curve. Used much in font definitions.

MOVETO

As the above commands specify the end-point but not the start-point, a method needs to be supplied to move to a new start-point without drawing (milling) a line. The Moveto command makes such positioning move.

DeskProto will also import CAD-layer information (if present). When selecting curves you can make each layer visible or invisible.

Postscript also may contain many other entities, like colors and bitmaps, which will be skipped by DeskProto.

Viewing the Vector data
After loading, you can see the Vector data on your graphics screen.
In case you do not see the curves, check if item "Vector curves" is checked in the Items visible dialog.

5.3 Geometry Data

DeskProto can load three types of **CAD data**: Vector data, **Geometry data** and Bitmap data.

For Geometries DeskProto only recognizes surface data defined with triangles: the outer surface of the geometry described using a (large) number of triangles. This type of geometry is also called Polygon data, and any 3D shape can be defined this way. *DeskProto will generate the toolpaths by projecting a flat toolpaths pattern over the 3D geometry.*
It is not the most efficient way of storing 3D data, still it has the large advantage that it always works: the definition is so basic that no incompatibilities between systems exist. This same format is also used for 3D printing, and because of that it can generated by any 3D CAD system

In order to create Geometry toolpaths you need to use a Geometry Operation in your project.

How to load a Geometry file
Geometry data files can be loaded in several ways:

- File menu command Load Geometry file
- Toolbar button Load Geometry file
- Button Add... on the Geometry tab of the Project parameters
- The Wizard Basic Geometry machining and other wizards.

It is possible to load more than one Geometry file, using the same command: after you have loaded a geometry the Load Geometry file command will change to Add Geometry file.

How to save Geometry data
Geometry data can be exported in two ways:

- File menu command Save Geometry data as...
- Button Save as... on the Geometry tab of the Project parameters

This will save all Geometry data to one file (also when various Geometry files had been loaded).
You can select any of the supported Geometry file types.
When saving the Geometry data you may select which of the Part transformations you want to apply for the exported file. You can do so in the Save Geometry Options dialog.

Geometry File types
DeskProto supports three geometry file types: STL, DXF and VRML. The only type that is fully supported is STL, as that file can only contain triangles. The other types may also contain other shapes (entities), which are not supported.
For loading geometry we advise to use **STL** files, as both DXF and VRML in practice may give problems.

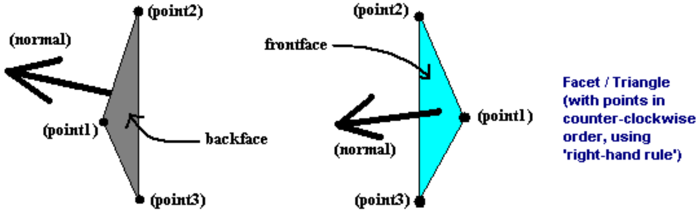
Two other file types that are also used to transfer geometry data are **not** supported by DeskProto: **IGES** and **STEP**. These formats store the data in a much more complex way, making many types of conversion errors possible

STL

File format created for the first 3D printer ever: the **ST**ereo**L**ithography system. This does well explain the acronym, though some claim that it stands for **S**tandard **T**riangle **L**anguage, or for **S**tandard **T**esselation **L**anguage..
An STL file may be a binary file or an ASCII file, the only difference being the storage method: the contents of both files (the series of triangles) is identical. DeskProto can read and write both binary and ASCII STL files.

The original StereoLithography system had more severe requirements for the **STL file** than DeskProto. It accepted only positive coordinate values, and one complete and true solid needed be present as geometry (no **gaps** and/or **orphan surfaces**). Currently negative coordinates no longer are a problem, however for 3D printers small gaps (say 0.001 mm between two triangles) still are lethal. DeskProto does not care about all these errors.

In an STL file for every triangle also a normal-vector is saved, in order to define the inside and the outside of the part. When the option 'Skip Backfaces' in the Project parameters has been checked the backsides of the triangles will not be considered: drawn in black, and skipped in the toolpath calculations.
This normal vector contains in fact double information, as the order in which the three points are stored can also be used to define inside and outside.



DXF

AutoCAD **D**rawing e**X**change **F**ile. A DXF file can contain many different entities, both 2D and 3D.
DeskProto only supports a specific type of geometry in the DXF file: geometry in small triangles (facets). In DXF these supported entities are called 3D Face and Polyface Mesh. All other entities in the DXF file will be ignored by DeskProto:

3D-Face

With 3D-faces for every triangle 3 points are stored. No normal-vector is stored, the three points are stored in counter-clockwise order.

Polyface Mesh

With polyface meshes first a long list of points is stored, and then for every triangle 3 indexes that each lead to a point in that list. In this way all the points only have to be saved once, thus saving disk-space.

Other entities in the DXF file will be skipped when importing the file. In practice many DXF-files will **only** contain other entities, and thus can not be read by DeskProto.

Note that DeskProto also supports DXF files for Vector data, and then will accept a different set of entities.

VRML

Virtual **R**eality **M**odeling **L**anguage file. Not widely used.
The standard for VRML files is almost completely covered by DeskProto, both for VRML Version 1 and Version 2. DeskProto does not support the entities Sphere, Cone and Cylinder, and the concept of Custom Node Type definitions.
VRML files have the file extension .WRL (short for "world").

Corrupt geometries

Unfortunately not all 3D CAD systems will export perfect geometry files, which may result in geometries that do not look right on the DeskProto screen. Note that with an up-to-date CAD program this is not likely to happen: most current CAD systems have a correct STL export function.

A known problem with the STL files of some surface modeling CAD programs is incorrect normal-vectors. The normal-vector of a triangle defines which side of the triangle is the outside, so incorrect normals result in a geometry with the **backfaces** on the outside, drawn in black. This can be solved by switching the option **Flip normals** on in the Project Parameters dialog.

You can also save your geometry with these flipped normals: choose the Save Geometry As option in the File menu.

STL-files contain normal-vectors.

DXF files do not contain normals, when importing a DXF file DeskProto calculates the normal based on the three points of the triangle being stored in a counter-clockwise order.

When some of the normals are OK and some are incorrect, then only some of the triangles are drawn in black.. This can also be solved by DeskProto. You have to uncheck the option **Skip backfaces with calculations** in the Project Parameters dialog. This causes all the triangles to be treated as if they don't have backfaces but 2 frontfaces so they will be rendered from both sides, and both sides will be used for toolpath calculations as well.

Obviously files can be corrupt in many more ways. DeskProto is very tolerant here: it will just accept files containing cracks, holes, orphan surfaces and many more inconsistencies. Files that would be rejected by any other RP-system.

Viewing the Geometry data

After loading, you can see the Geometry data on your graphics screen. As a wireframe, as a wireframe with hidden lines removed, as a rendered geometry or as points. To determine how to view the geometry you can check or uncheck the items in the Items visible dialog.

5.4 Bitmap Data

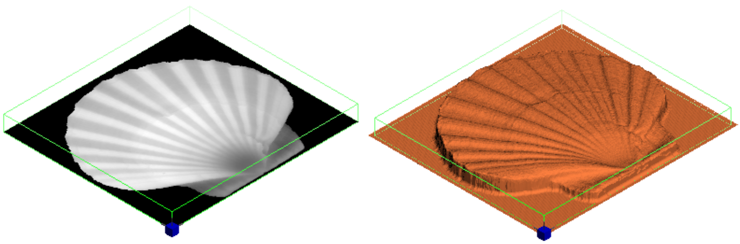
DeskProto can load three types of **CAD data**: Vector data, Geometry data and **Bitmap data**.

A Bitmap file contains a 2D picture by storing the color value of each pixel on the screen. So a line is stored as a series of black pixels on a white background. This in contrast to a vector file, where the line is stored using the coordinates of both the start point and the end point of the line. As many different colors can be used, in a bitmap complex pictures are possible. Digital photos for instance are stored in bitmap files.
DeskProto needs an image in gray values, so it will automatically convert color pictures to Black&White (B&W) when loading. In case you are interested: the conversion formula is *gray-value = 0.30 * red + 0.59 * green + 0.11 * blue.*

A bitmap file is a 2D file: a pixel only has an X-coordinate and a Y-coordinate, plus of course a color (gray value). DeskProto will convert this 2D file to a 3D geometry by translating the gray value to a Z-value, resulting in XYZ information for each pixel. The result is a 3D Relief. This is called **Gray scale to Z-height conversion**, or **Bitmap to Relief conversion**

The B&W 'colors' range between black and white, where pure black and pure white define the extreme Z-values to be used. These two extreme Z-values can be set in the Part parameters, tab Z Settings. Internally DeskProto converts the bitmap into a Z-grid, that can be used for toolpath calculations.

In order to create Bitmap toolpaths you need to use a Bitmap Operation in your project.



The two pictures above show the original bitmap image of a shell left, and a simulation of the resulting 3D Relief. Minimum Z has been assigned to black, and Maximum Z to white. DeskProto does not display the resulting Bitmap geometry, you can only see it by displaying the Z-grid, the Toolpaths or the Simulation.

Important to know is that for large bitmaps DeskProto will display a simplified version (256x256 pixels) of the bitmap, in order to speed up drawing. The actual toolpath calculations will nevertheless be done with the original bitmap data.

How to load a Bitmap File

Geometry data files can be loaded in several ways:

- File menu command Load Bitmap file
- Toolbar button Load Bitmap file
- Button Browse on the Bitmap tab of the Project parameters
- The Wizard Basic Bitmap machining and other wizards.

You can only load one bitmap file per project.

How to save Bitmap data

Geometry data can be exported in two ways:

- File menu command Save Bitmap data as...
- Button Save as... on the Bitmap tab of the Project parameters

You can select any of the supported Bitmap file types, except GIF..

Bitmap File types

DeskProto supports the five most popular file types for bitmap files:

BMP	File format defined by Microsoft for **B**it**M**a**P** files. Not very efficient (large files), widely accepted.
GIF	the **G**raphic **I**nterchange **F**ormat is an efficient format without data loss, though only max 256 colors are possible.
JPG	or JPEG File format defined by the ISO (International Standards Org) **J**oint **P**hotographic **E**xperts **G**roup. Various levels of compression are possible, all resulting in some loss of information.
PNG	the **P**ortable **N**etwork **G**raphics is the best bitmap format: with compression, no data loss, not patented (like GIF). Designed for Internet use.

TIFF **T**agged **I**mage **F**ile **F**ormat, originally made by Aldus, since 2009 controlled by Adobe. TIFF files tend to be very large.

Grid complications: the Moiré effect.
In a DeskProto bitmap operation two grids are used: the bitmap is a grid, and for calculating the toolpath the Z-grid is used. Be warned: combining two grids that are not equally spaced may lead to ripples in the resulting relief, caused by a Moiré effect.

The bitmap grid is the grid of pixels, the size of each grid cell in mm being set on tab XY Transform of the Bitmap parameters.
The Z-grid is the rectangular grid of XY positions, with a Z-value calculated for each position, the size of each grid-cell in mm set by the Precision.
The **Z-grid** will be filled with Z-values that are calculated in the **Bitmap grid**.

In most cases the dimensions of these different grid cells will not be not equal, so this conversion cannot be done one to one.
For instance: imagine a bitmap with a pixel size of 0.9 mm and a Z-grid with a cell size of 1 mm. Most Z-grid cells then are filled with the Z-value of just **one** single bitmap cell, however every 10th cell will be filled with Z-values from **two** bitmap cells. And this may cause a visible ripple in the relief. Such ripples are called the Moiré effect.

In order to prevent such Moiré problems you can select a pixel size that is dependent from the precision. This can be accomplished using one of the two options “Calculate from precision...”. The edit fields for Custom will show the resulting relief size. Note that when you later change the precision this will automatically also change the size of the resulting relief.

Viewing the Bitmap File data
After loading, you can see the Bitmap image on your graphics screen.
In case you do not see a bitmap, check if item “Bitmap” is checked in the Items visible dialog.

5.5 Project

A project is the object that you open when you work with DeskProto, it can be compared with a document in applications like MS Word. Unlike Word, where you can open several documents at the same time, in DeskProto you can have only one project opened. A project can be saved and opened in the File Menu, as a **D**eskProto **P**ro**J**ect file with the file-extension **DPJ**.

Project Structure
The information in a project is structured: in addition to some general information three groups are present:

- General information
- CAD files
- Parts
- Operations

Most important general information is the **Machine** (CNC milling machine) that you want to use for this project. You can select a different machine in the Project Parameters dialog. Other general information includes the type of user interface that you used (Wizard-based or Dialog-based) and the Edition of DeskProto that was used.

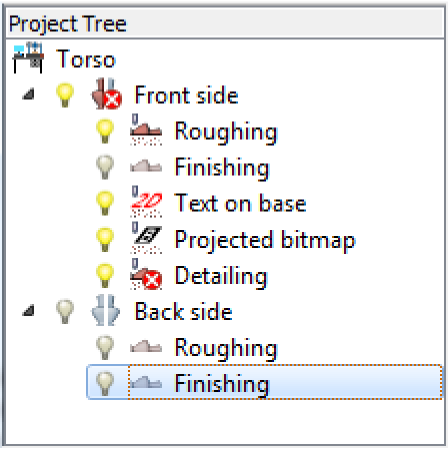
Each project can include one or more **CAD files**. Three types of CAD data are supported: Vector data, Geometry data and Bitmap data.
(in the DPJ file only links to the CAD files are stored, not the actual CAD data). In case a project does not have link to a CAD file it is called a template project.

The **Part** is what you are going to machine. A project can contain more than one part (zero parts is not possible), as for one model several parts may be needed. For instance the left halve and the right halve of a hand drill. The part definition consists of a number of parameters, like scaling values (relative to the original geometry), rotation values etc. Each type of CAD data comes with slightly different set of part parameters: the Vector settings, the Geometry settings and the Bitmap settings.
In fact the **Part parameters define what to machine**.

An **Operation** defines how the material will be machined. Three different types of operation are available: Vector operations, Geometry Operations and Bitmap operations. Each part contains one or more operations (no upper limit, zero operations is not possible). It is called operation because it defines

one actual machining operation. This definition also consists of a number of parameters, like milling direction, precision values, feedrate etc. Most times you will use more than one operation for one part. For example you may want to quickly remove most of the material (roughing) before you want to mill precisely (finishing).
In fact the **Operation parameters define how to machine it**.

In DeskProto the structure of the project is shown in the Project tree, which is placed on the left of the screen and looks like this:



Project Types
You can create a new project by choosing the New Project option in the File menu. You will see three sub-options for the three types of project that DeskProto supports:

- New **Vector project** will create a project with a Vector operation
- New **Geometry project** will create a project with a Geometry operation
- New **Bitmap project** will create a project with a Bitmap operation

So the difference is small: only the type of operation that is present, after which you can easily add and remove other operations, of any type.
When you create a new project the currently opened project file will be closed. In case you start DeskProto a new project will be created automatically.

Default project

When a new project is created, all parameters are copied from the Default Project, the Default Part and a Default Operation. This default operation can either be Vector, Geometry or Bitmap. The three operation types each have their own set of default settings: the default Vector operation, default Geometry operation, the default Bitmap operation.
All these settings are saved on your PC (in the registry), every user has a separate set of defaults. You can use these defaults to enter parameters that you want to use as a standard, for instance the **machine** that you always use. A default CAD file cannot be set though, and the default project can only contain one Part.
One of the settings of the default Part is which type of operation needs to be loaded (Vector/Geometry/Bitmap). This defines whether the default for a New project (in the file menu) is a Vector project, a Geometry project or a Bitmap project.
Editing the parameters of the default Project/Part/Operation can be done in the Options menu.

How to open a project

You can open a project by choosing the Open... option of the File menu. When you open a project the currently opened file will be closed.
The current folder when you first open the Open File dialog will be initiated conform the file location 'Data', which can be changed at the General tab page of the Preferences dialog. After that DeskProto will remember the folder you are working is (the current folder) and will keep using that.

How to save a project

You can save a project by choosing the Save option of the File menu. Using this option it will be saved under the same name you have saved it before. If you want to save it with another name, choose the option Save As... under the File menu.
The same current folder as just explained for Open project will be used here as well.

The DPJ file

The project information is stored in a **D**eskProto **P**ro**J**ect file (file extension **.DPJ**), containing all Parameter settings and also all View settings. Note that the project file does not contain any CAD data, only links to the CAD file(s).

Two different types of DPJ files are used:
1 - a project file with only the settings (default)

2 - a project file with both the settings and the calculated toolpaths.
Both files have the same file extension .DPJ

The second type (**with toolpaths**) can be used in case the toolpath calculations take very long.
You can choose between both types in the "Save as type" field of the Save As dialog.
When opening a project file it is not needed to distinguish between both types of DPJ file.
In fact the toolpaths will be stored in a separate file: the **D**esk**P**roto **T**oolpath file (file extension **.DPT**), which will be much larger than the (small) DPJ file.
For a project file called test.dpj the toolpath filename will be test.dpt
The difference between such Toolpaths file and an NC program file is that the first is in a machine-independent format.

A DPJ file without CAD data is called a template file: when opening DeskProto will ask for a CAD file to be loaded.
The sample DPJ files do not include a line to define the machine to be used: DeskProto will then use the default machine that you have set.

5.6 Part

A part is what you will machine using one setup (a block being fixtured on the machine). Many models for instance can easiest be created by milling two separate Parts, for instance the left side and the right side of a hand drilling machine, to be glued together later. In case you machine one block of material from two (or more) sides, DeskProto will see each side as a separate Part.

The description of a part consists of a number of parameters, like scaling values (applied to the original CAD data), rotation values and the size of the material block to be used. These parameters depend on which type of CAD data has been loaded: the **Vector settings**, the **Geometry settings** and the **Bitmap settings** are slightly different. For a complete list of parameters see Part Parameters dialog. To edit a part go to the Parameters menu and select the item Part Parameters. This opens the Part Parameters dialog.
A much quicker way to open the Part parameters dialog is double-clicking on the part's name in the Tree.

How the part should be machined is defined in its Operations. A part can contain one or more operations: Vector operations, Geometry operations and/or Bitmap operations.

Current part

In case your project contains more than one part you need to make a part **current** in order to see it on the graphics screen. There is always one part that's current: no more, no less. The light-bulb icon in the project-tree shows which part is current: for only part the light burns (yellow) - for all other parts (if any) the light is off (gray).
You can make a part current by clicking on the gray light-bulb icon for that particular part. Clicking on a yellow light-bulb does not have any effect. You can also make a part current by right-clicking on a part in the project-tree and then mark the option 'Make Current' in the context-menu that will be shown.

Default part

When a new project is created it will contain one part. The parameters of that part are copied from the default part. The default settings are saved on the computer (in the registry); each computer user has his/her own set of defaults.

You can use the default part to enter the settings that you want to use as a standard. For instance when your geometry always needs to be rotated 90 degr round Z, or when you always need the Zero point to be at the bottom of the block. You can also set the number of operations, for instance a Roughing operation and a Finishing operation (in that case you will have two default operations).

Important is the type of Operation that you use in your default part:

- when this is a Vector operation your default project will be a Vector project
- when this is a Geometry operation your default project will be a Geometry project
- when this is a Bitmap operation your default project will be a Bitmap project

To edit the parameters of the default part, go to the Options menu and select the option Default Part Parameters. This opens the Part Parameters dialog. The parameters of the default part are also used when you add a new part to the project.

5.7 Vector Operation

An operation gives a description of how the material should be machined. DeskProto features three different operation types:

- this **Vector operation** is meant to create toolpaths for Vector data
- a Geometry operation is meant to create toolpaths for Geometry data
- a Bitmap operation is meant to create toolpaths for Bitmap data.

The description of a Vector operation consists of a number of parameters, like cutter, type of toolpath (Profiling, Pocketing, Drilling), milling direction, precision values, feedrate etc. For a complete list of parameters see Vector Operation Parameters dialog.

To edit an operation go to the Parameters menu and select the item Operation Parameters. This opens a dialog to select which Operation to edit. After selecting a Vector operation, DeskProto will show the Vector Operation Parameters dialog. Of course a Vector Operation must already be present in order to select one.
A much quicker way to open the Vector Operation parameters dialog is double-clicking on the operation's name in the Project Tree.

Visible operations
To view the data or the toolpaths of a particular Vector operation you need to make it **visible**. Of all operations of the current part 0, 1 or more operations may be visible at the same time. The light-bulb icon in the Project-tree shows if an operation is visible: the light burns (yellow) means visible, the light is off (gray) means invisible.
You can make an operation visible by clicking on the gray light-bulb icon for that particular operation, and make it invisible by clicking on it's yellow lamp icon. You can also make an operation visible by right-clicking on an operation in the project-tree and then mark the option ‘Visible’ in the context-menu that will be shown. Another way to make an operation visible is when you check an operation in the operation-list of the Items visible dialog. To actually see one of the items (curves, toolpaths) of the Vector operation you have made visible, the item needs to be checked as well.

Default Vector operation
When a new Vector operation is created, the parameters of the new operation are copied from the default Vector operation. The default settings are saved

on the computer (in the registry); each computer user has his/her own set of defaults.
DeskProto also contains a default project and a default part. When the default part contains a Vector operation (as first operation) the default project is a Vector project.
You can use the default Vector operation(s) to enter milling parameters that you want to use as a standard. For instance a specific tool and/or machining depth.

To edit the parameters of the default Vector operation, go to the Options menu and select the option Default Vector operation Parameters. This will open the Vector operation Parameters dialog.

5.8 Geometry Operation

An operation gives a description of how the material should be machined. DeskProto features three different operation types:

- this Vector operation is meant to create toolpaths for Vector data
- a **Geometry operation** is meant to create toolpaths for Geometry data
- a Bitmap operation is meant to create toolpaths for Bitmap data.

The description of an operation consists of a number of parameters, like cutter, strategy, precision values, feedrate etc. For a complete list of parameters see Geometry operation Parameters dialog.

To edit an operation go to the Parameters menu and select the item Operation Parameters. This opens a dialog to select which Operation to edit. After selecting a Geometry operation, DeskProto will show the Geometry Operation Parameters dialog. Of course a Geometry operation must already be present in order to select one.
A much quicker way to open the Geometry operation parameters dialog is double-clicking on the operation's name in the Project Tree.

Visible operations
To view the data or the toolpaths of a particular Geometry operation you need to make it **visible**. Of all operations of the current part 0, 1 or more operations may be visible at the same time. The light-bulb icon in the Project-tree shows if an operation is visible: the light burns (yellow) means visible, the light is off (gray) means invisible.
You can make an operation visible by clicking on the gray light-bulb icon for that particular operation, and make it invisible by clicking on it's yellow lamp icon. You can also make an operation visible by right-clicking on an operation in the project-tree and then mark the option ‘Visible’ in the context-menu that will be shown. Another way to make an operation visible is when you check an operation in the operation-list of the Items visible dialog. To actually see one of the items (Z-grids, toolpaths) of the Geometry operation you have made visible, the item needs to be checked as well.

Default operation
When a new Geometry operation is created, the parameters of the new operation are copied from the default Geometry operation. The default settings are saved on the computer (in the registry); each computer user has his/her own set of defaults.

DeskProto also contains a default project and a default part. When the default part contains a Geometry operation (as first operation) the default project is a Geometry project.
You can use the default Geometry operation(s) to enter milling parameters that you want to use as a standard. For instance a specific tool and/or strategy.

To edit the parameters of the default Geometry operation, go to the Options menu and select the option Default Geometry operation Parameters. This will open the Geometry operation Parameters dialog.

5.9 Bitmap Operation

An operation gives a description of how the material should be machined. DeskProto features three different operation types:

- this Vector operation is meant to create toolpaths for Vector data
- a Geometry operation is meant to create toolpaths for Geometry data
- a **Bitmap operation** is meant to create toolpaths for Bitmap data.

The description of a Bitmap operation consists of a number of parameters, like cutter, strategy, milling direction, precision values, feedrate etc. For a complete list of parameters see Bitmap Operation Parameters dialog.

To edit an operation go to the Parameters menu and select the item Operation Parameters. This opens a dialog to select which Operation to edit. After selecting a Bitmap operation, DeskProto will show the Bitmap Operation Parameters dialog. Of course a Bitmap Operation must already be present in order to select one.
A much quicker way to open the Bitmap Operation parameters dialog is double-clicking on the operation's name in the Project Tree.

Visible operations
To view the data or the toolpaths of a particular Bitmap operation you need to make it **visible**. Of all operations of the current part 0, 1 or more operations may be visible at the same time. The light-bulb icon in the Project-tree shows if an operation is visible: the light burns (yellow) means visible, the light is off (gray) means invisible.
You can make an operation visible by clicking on the gray light-bulb icon for that particular operation, and make it invisible by clicking on it's yellow lamp icon. You can also make an operation visible by right-clicking on an operation in the project-tree and then mark the option ‘Visible’ in the context-menu that will be shown. Another way to make an operation visible is when you check an operation in the operation-list of the Items visible dialog. To actually see one of the items (bitmap, toolpaths) of the Bitmap operation you have made visible, the item needs to be checked as well.

Default Bitmap operation
When a new Bitmap operation is created, the parameters of the new operation are copied from the default Bitmap operation. The default settings are saved on the computer (in the registry); each computer user has his/her own set of defaults.

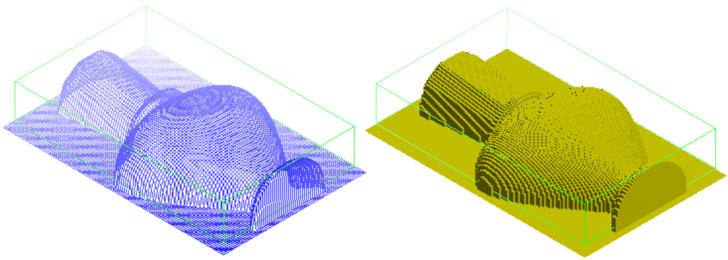
DeskProto also contains a default project and a default part. When the default part contains a Bitmap operation (as first operation) the default project is a Bitmap project.
You can use the default Bitmap operation(s) to enter milling parameters that you want to use as a standard. For instance a specific tool and/or strategy.

To edit the parameters of the default Bitmap operation, go to the Options menu and select the option Default Bitmap operation Parameters. This will open the Bitmap operation Parameters dialog.

5.10 Z-Grid

The Z-grid is an intermediate result, in-between the CAD-data and a toolpath. It is used for Geometry data and for Bitmap data: Vector data is processed without a Z-grid. The Z-grid is a sort of height-map of the geometry (or of the bitmap relief): a 3D bar graph, with a Z-value for each XY position. The size of the grid-cells is set by the toolpath distance and the stepsize along the toolpath. Normally you should not be bothered by this representation of the geometry, however it might be useful to see what is really happening. For instance in case a hole in the geometry was missing in the toolpath, you could check the Z-grid. In case present there, the cause will be that the cutter is too large to fit in the hole. In case not present in the Z-grid, some error is present is the geometry file. Besides that it can also give an idea of how the final result will look: a rough simulation of the part to be created. Especially when you view it as a rendered Z-grid.

Here's an image with a Z-grid and the rendered one (Note that a rendered Z-grid for an accurate toolpath needs some time to be drawn on your screen).



The pictures show a line-drawing and a rendered Z-grid, you can display these by checking them in the Items visible dialog. The grid-structure is clearly visible. Note that any undercuts present that are present have become solid: just one Z-value for each grid-cell. Especially for inverse milling this rendered Z-grid is useful as a preview to show what will be created. Also note the strong staircase effect that is visible. In the actual model the stairs will be partially smoothed away due to the size of the cutter (as it cannot create sharp inner corners).

Each operation has it's own Z-grid.

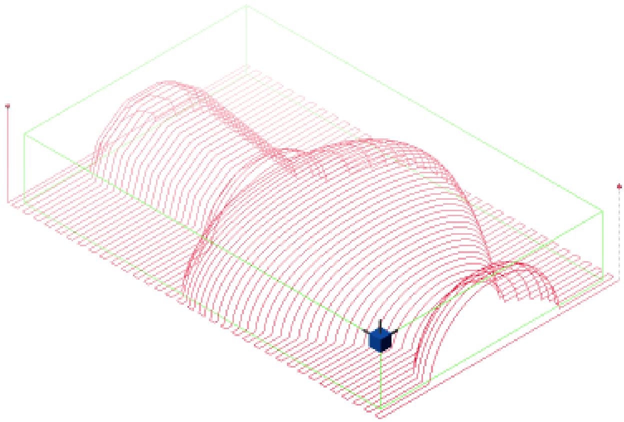
How to calculate and show a Z-grid

You can show a Z-grid by selecting the option Calculate Z-grids under the Create - Extra menu. Another way is checking Z-Grids or Rendered Z-Grids in the Items visible dialog.

5.11 Toolpath

The Toolpath is the path that the cutter will follow to create a part on the milling machine. More specific, it is the series of positions for the **tip** of the cutting tool (XY of the center of the tool, Z of the lowest point of the tool). The start of the toolpath is drawn as a small red cone pointing downward, the end by a similar cone pointing upward. These cones are NOT the workpiece zero point: in the Items visible dialog an option is present to display a blue orientator at the workpiece zero point.

The toolpath is a very important representation for a visual check before starting the milling machine: any possible errors should be found before milling. Here's an image of a toolpath.



Most lines are drawn as solid lines (default in red), indicating a normal speed (Feedrate).
Lines drawn in purple indicate that the feedrate is reduced by DeskProto's Dynamic Feedrate Control.
Some lines are dashed in gray, indicating that they are **Rapid** movements (moving as fast as possible).
The toolpath depends on the settings made in the Operation Parameters dialog.

After calculating a toolpath using the command **Calculate toolpaths** the toolpath will automatically be switched on in the scene (made visible in the Items visible dialog). To send this toolpath to the machine it should be post-processed to an NC-program, which is done using the command **Write NC-program file**. How it will be post-processed is determined by the postprocessor which is linked to the milling machine you use.

How to calculate toolpaths

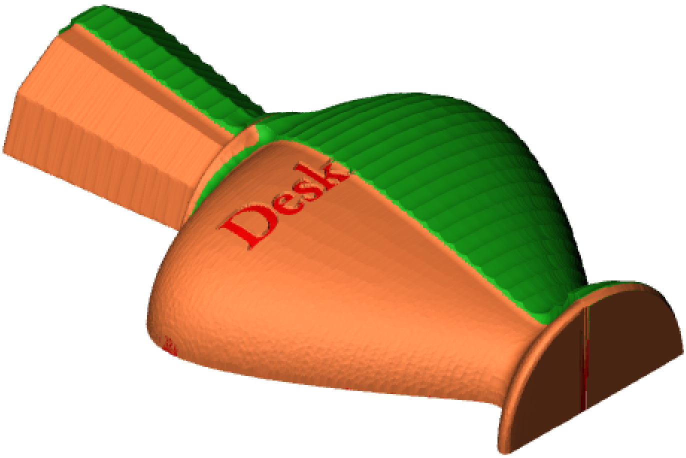
You can let the toolpaths be calculated by selecting the option Calculate Toolpaths under the Create - Extra menu.

As a shortcut this convenient button is present on the Toolbar.
Another way is checking the Toolpaths checkbox in the Items visible dialog.

5.12 Simulation

The NC program file is in fact the final result of DeskProto: sending it to the machine will produce the desired part. It may be useful though to preview this resulting model on screen, in a machining Simulation.

The Simulation is a drawing on screen that shows you what the resulting part will look like. This can be used to check things like the resulting surface smoothness, error movements (if any) that damage the part, rest material where the cutter cannot reach, etc. DeskProto will calculate a simulation in 3D, so you can rotate, pan and zoom it just like any other item on screen.

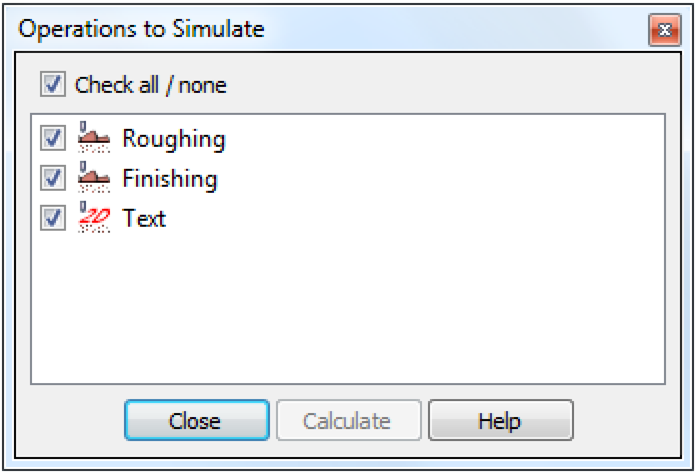


The simulation is for one Part: all available operations can be included. The illustration above shows three operations:

- a Geometry operation for roughing, where the ballnose toolpaths (or rather the "cusps" in-between) are clearly visible: the green part on the right.
- a Geometry operation for finishing, showing a very smooth resulting surface: only the left half has been finished.
- a Vector operation with a DeskProto logo, 0.5 mm deep, projected onto the bottle surface: only visible on the finished side.

In a simulation an operation cannot simply be made invisible (like for instance for the toolpaths):
just as in real life for a simulation removing material is possible while undoing such removal is not.

Because of that the Operation's visibility settings cannot be used here, and a Simulation comes with the Simulated Operations dialog to select which operations need to be included.
Only after pressing Calculate the simulation will be updated (if needed after resetting it first).



You can set the **Level of detail** of the simulation on the Simulation tab of the Part parameters.

For Geometry projects DeskProto simulation allows you to **compare the simulation with the original geometry**, and indicate any differences with a color. Rest material in green, too much material removed in red. On the Simulation tab of the Part parameters the user can select whether or not to use these colors, and also the tolerance to be used.
In the simulation screenshot above you can for instance see that a skin has been applied when roughing (green rest-material), and that the cutter is too thick for the small inner radius at the neck of the bottle (green rest-material just below the cap). The red color in the DeskProto logo is logical, as the logo is not a part of the geometry and thus DeskProto concludes that too much material has been removed. The red will only be visible in case the geometry is switched off in the Items visible dialog.

How to calculate and show a simulation
You can show a Simulation by selecting the option Calculate Simulation in the Create menu.

As a shortcut this convenient button is present on the Toolbar.
Another way is checking Simulation in the Items visible dialog.
The speed of drawing a simulation can be optimized using the Graphics options in the Advanced Preferences.

5.13 NC-program

NC stands for Numerical Control (we will also use the acronym CNC, which stands for Computerized Numerical Control). An NC-file contains an NC program: a series of instructions for a CNC milling machine to execute. It needs to be sent to a machine to make that machine execute the NC program. Each file contains commands for a specific machine (or machine-type).

An NC-program is machine-dependent: its format will be different per machine, as each machine (or rather each controller) will "speak a different language". The format that DeskProto uses to write the NC-file is determined by the Postprocessor that is configured for that machine. And in the Project parameters you can select which machine will be used.
Almost any NC program file is in plain ASCII: you can read and edit it using a plain text editor like Notepad. DeskProto can only work with ASCII NC files.

How to create an NC-program
The toolpaths calculated by DeskProto can be saved to an NC-program choosing the option Write NC-program from the Create - Extra menu.

As a shortcut a convenient button is present on the Toolbar.

DeskProto will try to combine the toolpaths of all Operations of one Part into one combined NC program file.
Or, if Chaining has been applied, even toolpaths of Operations in several Parts can be combined.

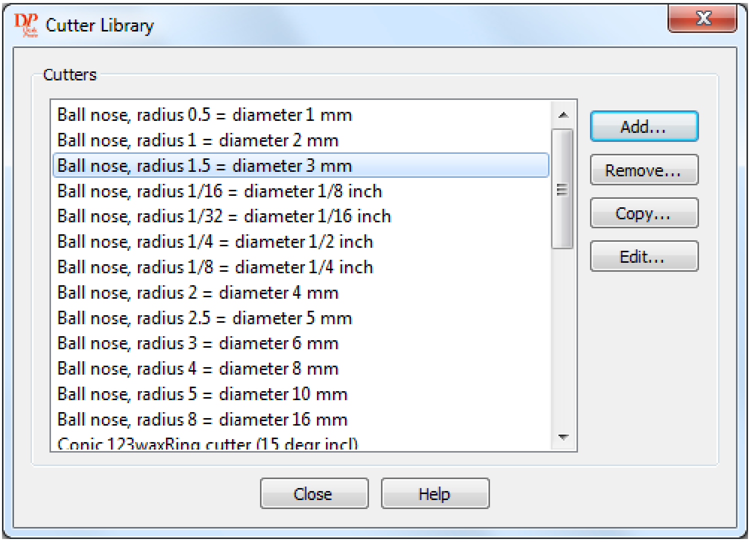
NC Files list
DeskProto can show you a list of all NC files that have been written for the current project. This is shown in the NC Files window, below the Project Tree. If this window is not visible you can check it in the View menu.

Sending NC-program to machine
For some machines it’s possible to send an NC-program to the machine from within DeskProto. How to do this see the option Send NC-program To

Machine from the Create menu. For all other machines you need to open the NC file in the machine's control software.

5.14 Libraries

Three different libraries are present in DeskProto: one for machines, one for postprocessors and one for cutters.
Each library is displayed in a dialog like this:



A library is just a collection of definitions. You can either add, copy, remove or edit an item. The definitions are saved to disk in the file location Drivers, which can be set at the General tab page of the Preferences dialog box. They are saved to disk when you click the OK-button of the library-dialog, one file for each driver.

When you add or copy an item, you can set all the values including the filename. When you edit an item you can set all values except the filename. If you want to change the filename of a cutter, postprocessor or machine you can use Windows Explorer to do so. You need to do this when DeskProto is not running (so close DeskProto first) because otherwise the old filename will be saved again when closing the library. The file with the new name will be automatically loaded when you restart DeskProto.

See Library Of Machines, Library Of Postprocessors and Library Of Cutters.

All these definitions are stored in files (machines in .mch files, postprocessors in .ppr files and cutters in .ctr files), to be stored in the Drivers directory of DeskProto. All these files are plain ASCII text files, that can also be edited using a plain text editor.
For editing no help information is present, so only do this if you are sure what you are doing.

5.15 Two-sided machining

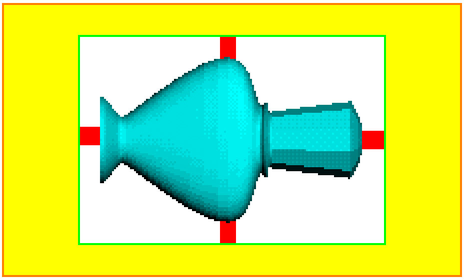
This issue is about Geometry machining (does not apply to Vector and Bitmap).
Some models can be completely machined from one side, like DeskProto's DpPictureFrame sample geometry.
For other models, like the perfume bottle sample geometry, machining from one side is not sufficient.
A complete bottle can be created in three ways:

1. Machine two separate halves and glue these to one another to make a complete model. This is the easiest way, which works great for many types of models. For instance for an electric hand-drill or similar tool.
2. Use a rotation axis to rotate the model during machining. This is called rotation axis machining.
3. Machine the model from two sides, so flip the block upside-down halfway through process. This flipping can be done either manually or automatically by a rotation axis.

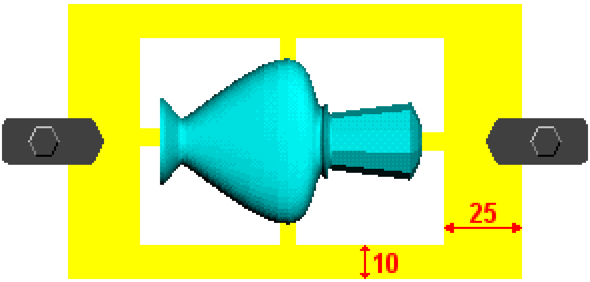
This help page concerns **Two-Sided machining** using the manual flip, for the automatic flip see the page on indexed machining.

Machining from two sides comes with some extra issues:
- how to hold the part after turning it upside down
- how to make sure that both sides are perfectly aligned
- how to correctly set the WorkPiece zero point for the second side, making it match the first side.

The **Two-Sided milling** **wizard** (manual flip) will help you to create a complete model in one piece, by machining it from two sides. The total block of material to be used is larger than the model, leaving a frame around the model. This frame permits you (after machining the first side) to again clamp the model in order to machine the second side as well (turned upside down). See the illustration below, where the yellow part is the block, or in fact what has remained after milling (top view).



Obviously the model needs to remain connected to the block when the second side is machined. This is done by the so called Support Tabs: **four** small rectangular blocks that function as bridges to connect the model to the remaining part of the block. In the illustration above the four red rectangles are the Support Tabs.
DeskProto will lock the Operation's Area to be machined to the geometry plus the Support Tabs by setting it to Custom, and DeskProto will switch off the Borders to prevent the tool from cutting the outside surface of each support tab (as that would disconnect it from the main block).

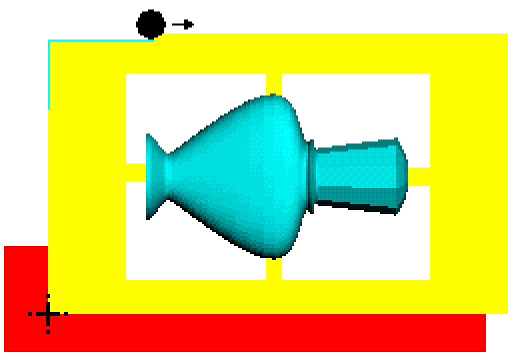


DeskProto will set the width and depth (X and Y) of the material block. For X both left and right 25 mm (or 1 inch) is added, permitting you to clamp down the block on the machining table at that spot. For Y both front and back 10 mm (or ½ inch) is added to end up with a block that is sufficiently stable when machining. These dimensions need not be very accurate. A bit oversize is advised for the actual block, making sure that you can machine a reference plane later in the process.

In contrast, the height of the block (Z) needs to be very accurate as otherwise the second side will not be correctly positioned relative to the first. In practice you can easily satisfy this requirement, as you can buy most materials in slabs at an exact thickness. Of these slabs also both sides are exactly parallel to one another.
The size of the total material block is defined in the Part parameters.

Note that DeskProto can be configured for two different block types: **floating** size and **fixed** size. The first type is default, and determines the block size by adding a certain frame thickness on all four sides, as explained above. The second type uses, as its name indicates, a fixed block size. This is handy when you have similar products and a stock of equally sized blocks. In this case it is possible that the part does not fit in the block (some minimum margin is needed as a frame): in such cases a warning message will be issued and you need to scale down the part. The choice between both types of block, and the actual dimensions, can be set in the Preferences dialog.

DeskProto makes it easy for you to correctly position the two sides relative to one another. Or in other words: to set the correct workpiece zero point to machine a second side that exactly fits the first. For this aim DeskProto uses two rulers (stop-bars) that are exactly parallel to the machine's axes, and have a known position. See the illustration below: the long 'horizontal' bar is parallel to X, at a known Y position, the short 'vertical' bar has a known X-position (being parallel is less important here). The workpiece zero point for both NC program files has to be set at the exact point where both rulers meet, with the Z=0 set with the tip of the tool touching the top of the block. This zero point remains the same after turning the block upside down for the second side. Also see the illustration below.



After machining the first side, you have to machine two reference planes, on the block’s back and left side: see the blue line in the illustration. The reference plane on the left is on position X = 0 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 specified in the **Report file**, and needs to be machined along the complete back. After turning upside-down these two reference planes will exactly touch the rulers, making sure that the block is exactly lined up with the machine, and exactly at the correct position.

Note:
The Ruler /Reference plane method just described is just one method: more methods are available to correctly position the block after turning upside-down. For instance using **reference pins** on the machining table, and drilling holes to exactly fit these positioning pins. This is easiest when you set the zero point exactly centered between the pins (two or four). In DeskProto "Make center of material zero" is a pre-defined option for the zero point X and Y (needs to be set after completing this wizard).
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.

The **Report file** just mentioned will be generated upon pressing the Create button, and gives all information needed for the actual machining from two sides. The file is opened in Notepad, making it easy to print the file and/or to save it. Default filename is Wizardlog.txt: do save it using a different name to prevent it from being written over by a next wizard.

Of course you are free to still fine-tune the results of the wizard before starting on the machine. You can still change any parameter after finishing the wizard: also parameters that have not been set by the wizard. You may also add operations, for instance some extra finishing with a small tool for some specific detail. In case of using an extra operation make sure to create the new operations by **Copying** an operation set by the Wizard, not by just Adding a (default) operation. That way all settings made by the wizard will be copied as well.
After changing any parameter you will have to again save the NC program file that was written by the wizard.

A wealth of information on two-sided machining is available in the DeskProto Tutorial book (can also be downloaded as PDF): Lesson 6 is about two-sided machining, and shows you how to machine a nice sample geometry: a cellphone's front cover. This STL file is a free download on www.deskproto.com

5.16 Rotation axis machining

A rotation axis or 4th axis on a CNC milling machine can be applied in two different ways:
- the part can be rotated during machining
- the part can be machined from one side (three axis machining), then rotated and machined from a second side, etc.

The first method is in fact 'real' rotation axis machining (**continuous rotation**), the second method is called **indexed machining**.
DeskProto supports both methods, obviously only in the Multi-Axis edition of DeskProto.
Indexed machining is possible only for Geometries

Rotation axis machining can be selected by checking the box "Use rotation axis" in the Part parameters.
When **Geometry** data is present DeskProto will then set a cylinder shape Material block that exactly fits the bounding box of the geometry. The axis of this cylinder will be the X-axis in CAD coordinates (the line for Y=0 and Z=0), so you may need the option Center geometry to center your geometry round that X-axis.
In case only **Vector** data and/or **Bitmap** data is present DeskProto will set a cylinder shape **Material block** that is exactly large enough to wrap the 2D CAD data round this complete cylinder (like a paper label on a jam jar). So for a bitmap image with Y-size 100 mm, the diameter of the generated cylinder block will be 31.83 mm: the cylinder's circumference is Pi * 31.83 = 100 mm.
For mixed projects the Geometry size will overrule the wrapping calculation.
In all cases you are of course free to change the block dimensions conform your needs.

For Geometries an easier way is available: use the Rotary machining wizard.

Rotation axis machining in DeskProto in fact still is a type of three axis machining: instead of moving X, Y and Z at the same time, now X, Z and A are moved. In a DeskProto NC program file for Rotation axis machining no Y-coordinate value is present.
So before starting the cutter needs to be positioned along Y exactly above the rotation axis, so at Y = 0.0. You can make DeskProto do this automatically by checking the option "*Write Y=0 in first movement command for rotary machining*" in the Advanced Machine Settings dialog.

DeskProto does not do this by default as such movement may damage your machine in case the current Z is too low.

A wealth of information on rotation axis machining is available in the DeskProto Tutorial book (can also be downloaded as PDF): Lesson 5 is about rotation axis machining a geometry, and shows you how to machine a great sample geometry: the head of the famous Venus de Milo statue. This STL file is a free download on www.deskproto.com

5.17 Indexed machining

A rotation axis or 4th axis on a CNC milling machine can be applied in two different ways:
- the part can be rotated during machining
- the part can be machined from one side (three axis machining), then rotated and machined from a second side, etc.

The first method is in fact 'real' rotation axis machining (**continuous rotation**), the second method is called **indexed machining**.
DeskProto supports both methods, obviously only in the Multi-Axis edition of DeskProto.
Indexed machining is possible only for Geometries

DeskProto offers indexed machining in the Advanced Geometry wizard **Two or more sides, automatic rotation** (also called the **N-Sided Milling** wizard). In this wizard you can choose any number for N, from 2 up to 99 sides.

The wizard will automatically generate N parts for you, and apply the correct rotation for the geometry in each Part. The rotations are evenly spread (360 / N), if needed you can select a higher N and then later delete some of the parts.
Support tabs (cylinder shaped blocks left and right) are optional.
In each part one or two operations are generated (finishing, and optionally roughing first): the wizard will link all these operations into one large Chain in order to write one combined NC file for the complete indexed milling process.
The required A-axis rotation commands for the rotation axis (in-between the operations) are entered in the operation's Start/End commands.

DeskProto also supports indexed machining on **five-axis** milling machines (five-axis continuous rotation is not supported). This works exactly the same as with four axes: you create a series of parts, specify the correct rotation for each part, and specify that same rotation in the Start commands of each Operation. Only now both an A-axis rotation command and a B-axis rotation command need to be specified.

The five-axis process however is much more complex than for four axis, as five-axis machines are available in many configurations. That is why DeskProto cannot offer a wizard to automatically create such project for you. A detailed instruction for five-axis indexed machining can be found in **Lesson**

9 of the DeskProto **Tutorial** book (can be downloaded as PDF). Including a nice sample project.

5.18 Graphically finding the rotation

Finding the correct XYZ rotation values to correctly orientate a Geometry as required can be very difficult, especially when angles other than 90 degrees are involved. Here is a tip to easily find these three rotation values by first rotating the geometry on screen.
Follow the following steps:

1. Set the rotation settings for the Part to [X=0, Y=0, Z=0].
(Part Parameters, tab page Transform)

2. Rotate the geometry to the required orientation (using the mouse) seen from above (assuming the X-axis of the machine to be horizontal on the screen, and the Y-axis of the machine to be vertical on the screen). So imagine that your viewpoint is above the machine: you are looking downward from the positive Z-axis.

3. Now simply copy the rotation settings from the view (View menu >> Viewpoint >> Custom) and enter them as the rotation settings for the part (Part Parameters, tab page Transform).

4. Finally check whether the rotations are exactly the way you want (by looking at some default views for example).

A different feature that you can use here is the option called Show downward faces in the Items visible dialog.

5.19 Editions / Trial mode

DeskProto is offered to you as **Freemium** software:
you are welcome to use the *basic functionality* DeskProto **free** of charge, the *advanced features* are available as **premium** extras.

Four different Editions are available:

- **Free Edition**
- **Entry Edition**
- **Expert Edition**
- **Multi-Axis Edition**

of which the first is free while for the other three you need to buy a license. The Free edition allows you to also **Trial** (evaluate) the higher editions: when running in **trial mode** the resulting toolpath will leave a **Trial cross** (watermark) on each part that is machined.

The **Free Edition** is available for anyone: free of charge, without any obligations attached. It's functionality is limited, still it offers all you need for

basic CNC machining: Profiling toolpaths based on Vector Data, Parallel toolpaths over Geometry Data, and machining reliefs based on Bitmap data. In the Free edition a project may contain maximum one part and one operation.
Many parameters as described in the Help file are not available in the Free Edition. Still the most important parameters are there, and for many users this free CAM program will be all they need.

The **Entry Edition** is the lowcost version of DeskProto, offering limited options, at a very low price.
A few important extra options are present (added to the Free edition): Pocketing and Drilling for Vector operations, and Roughing for all three operation types. Projects also may contain any number of parts and operations.

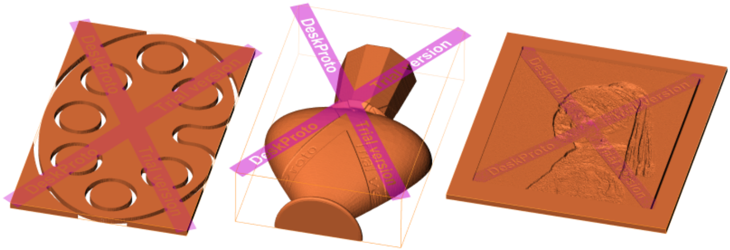
The **Expert Edition** includes all parameters, except for the rotation axis options. So the fourth axis and the fifth axis are not available in this edition.

The **Multi-Axis Edition** is the most complete version: all parameters are present, and so are the A-axis and B-axes rotation axes.

More information about higher editions can be found in the Upgrade dialog.
An edition comparison table can be found on www.deskproto.com

A license for a higher edition can be activated in the Activate dialog.
After activation your license this Edition select dialog will no longer be displayed.

5.20 Trial Cross



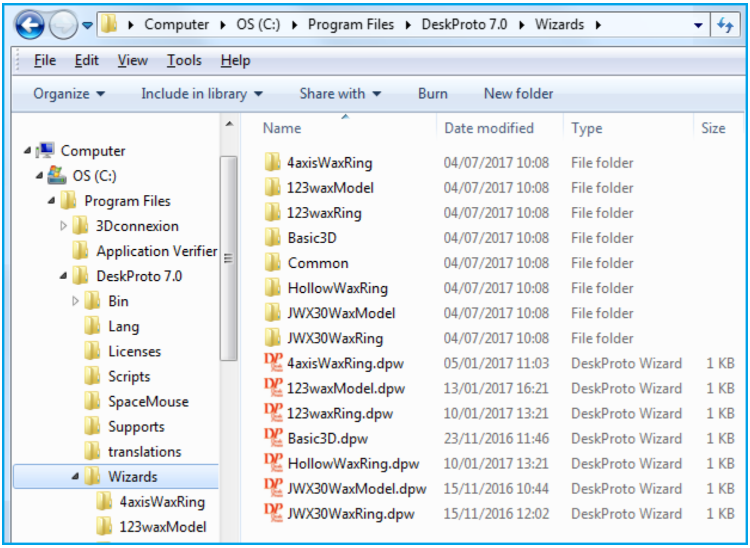
The Free edition allows you to **Trial** (evaluate) the higher Editions: when running in trial mode the resulting toolpath will leave a **Trial cross** (watermark) on each part that is machined.

The effect of this Trial cross (displayed in purple) can be seen in the above image:
For a **Vector** operation no toolpaths will be present in the area below this purple cross: the toolpaths will simply be interrupted.
For a **Geometry** operation and a **Bitmap** operation all Z-positions below this purple cross will be a lowered a bit, resulting in two "ribbons" pressed into the surface of the part, displaying the text "DeskProto Trial version".
So in all cases you can clearly see if the resulting part would suit your needs, however the trial cross will make really using the part impossible.

Note that the project files (DPJ files) that are saved when in trial mode do not have a trial limitation. So you can run in trial mode to prepare a project and then use a paid license on a different PC to calculate the NC file. For schools this makes a very lowcost classroom license possible: one paid educational license on the PC next to the machine, plus many free licenses running on the student PC's.

In order to use one of the higher editions you need to buy a license.
Such license for a higher edition can be activated in the Activate dialog.
After activating your license the trial cross will no longer be applied.

5.21 Custom Wizards



In addition to the series of predefined Wizards DeskProto also offers **Custom wizards**: wizards that any user can add and/or edit. A number of these custom wizards already has been installed, and can be started via File menu command Start Custom wizard.

The Custom wizard is a very powerful option, making it possible to add your own Wizards to DeskProto. Such wizard can make it easy to use a special machine, or to create a special type of product. A good example are the Wax Ring Wizards that come with DeskProto (123WaxRing, made for the 123WaxRing fixture, and HollowWaxRing, made for the 5t axis of the Roland JWX-10 milling machine). Total 7 custom wizards haven been installed by the DeskProto Setup, you are free to edit these wizards and/or use them as examples for your own script wizards.

A valid Custom Wizard consist of one DPW file (DeskProto Wizard) and a subdirectory with the same name.
For instance the file Basic3D.dpw and the directory *C:\Program Files\DeskProto 7.0\Wizards\Basic3D*
The DPW file specifies the name of the Wizard, how many pages are present, and the files that define each page. These files are QML files, present in the

folder just mentioned. For instance *C:\Program Files\DeskProto 7.0\Wizards\Basic3D\WizPageLoadGeometry.qml*

In this folder you may also use .QMLC files: compiled versions of the .QML files, used as cache for a better performance (faster display) of the wizard pages.

The wizard pages are written in **QML** (the Qt Modeling Language), which is a framework for developing applications offered by the QT Company. As DeskProto is built using the QT toolkit this is the most efficient choice for a framework. For each custom wizard page one QML file is needed. This file contains both the user interface (dialog design, texts, edit boxes, etc) and the script that defines the actions to be performed. As scripting language **JavaScript** is used. Both about QML and about JavaScript plenty of information is available via the Internet.

So basically all you need to add a Custom wizard to DeskProto is a plain text editor. Copy the resulting files to the location specified above, and DeskProto will show your new wizard. For copying to that location and for editing files at that location Windows requires administrative privileges.

For more information on scripting see the Scripts page and the **DeskProto Script Documentation** (email us to receive a copy). In August 2018 this documentation has been updated: email us in case you have an older version. Note that a very easy way to create a Script wizard is to copy and rename one of the sample Script wizards: then the correct structure already is present.

You can also start a Custom Wizard by calling it's DPW-file as a command line parameter.

For software specialists:
you can set the location of this wizard directory in the registry, at
HKEY_CURRENT_USER\Software\Delft Spline Systems\DeskProto\7.0\Preferences\File Locations\WizardsLocation
Do not change this unless you are sure what you are doing.

5.22 Scripts

A **Script** or **Macro** is a program: a series of commands for DeskProto that will be executed one by one. Programs are written in a programming language: DeskProto scripts need to be written in **JScript** (JavaScript, file extension .dps).
Other ways to automate DeskProto are Template projects and Command line parameters.

A script can be called in DeskProto using the command **Run Script...** in the File menu.
You can also start a Script by calling it as Command line parameter.
Default folder for script files is the Scripts subdirectory of DeskProto (C:\Program Files\DeskProto 7.0\Scripts)

A Script is an ASCII text file with a number of lines: each line contains a command or part of a command to be executed. The script can call a number of objects (functions) that DeskProto has made available via the Scripting Interface. Most of these objects are Properties (a variable that can be set or read) and Methods (a function with one or more parameters).

A very simple example of a DeskProto script file is present already (ScriptedBottle.dps):

```
var strSampleLocation = DeskProto.preferences.getSampleLocation();

DeskProto.project.loadGeometry(strSampleLocation + "Bottle.stl");
DeskProto.project.activePart.setRotation(-90.0, 0.0, 0.0);
DeskProto.project.activePart.segmentMethod = 2;
DeskProto.project.calculateToolpaths();
DeskProto.project.writeNCProgram( "ScriptOutputNCfile.ext" );
DeskProto.exit();
```

This script will load the Bottle geometry, orientate it correctly for machining, set the Material block (segment is the old name) to use the upper half only, calculate the toolpaths using the default Operation parameters, save these toolpaths in an NC program file and exit.
The parameter of function LoadGeometry could have contained the complete path in one string. However as the location of the Samples folder is different per Windows version (sorry: thanks to Microsoft) this solution is better.

The available scripting objects are described in the **DeskProto Script Documentation**: a series of HTML files that can be found in the directory \DPScript\html\ on the DeskProto CD. Open the file \DPScript\html\index.html to start reading. If you do not have this CD then please email us.
In August 2018 this documentation has been updated: email us in case you have an older version.

A special type of Script that is available in DeskProto is the **Custom Wizard**. This is a very powerful option, making it possible to add you own Wizards to DeskProto. Custom wizards can be started via **Start Custom Wizard** in the File menu.

For software specialists:
you can set the location of this scripts directory in the registry, at HKEY_CURRENT_USER\Software\Delft Spline Systems\DeskProto\7.0\Preferences\File Locations\ScriptsLocation
Do not change this unless you are sure what you are doing.

5.23 Command Line Parameters

It is possible to run DeskProto with one or two command line parameters. This feature is not meant for 'normal' users, but rather for application builders who need to include toolpath calculations in their application and/or want to automate that process. So when you do not know what a command line parameter is please ignore this paragraph.
Other ways to automate DeskProto are Template projects and Scripts.

Command line parameters for DeskProto in all cases except one (see below) need to be file names (valid file specifications, including the path). When the file is not found or invalid an error message will be shown.
For the first parameter only a limited number of file-types is possible (DeskProto uses the file-extension to decide what to do with the file), for the second parameter any file extension is possible.

1. First parameter:

Supported file-types for the first parameter are:
DPJ, DPW, DPS
STL, DXF, AI, EPS
BMP, GIF, JPG, JPEG, PNG, TIF, TIFF

DPJ
This is a **DeskProto project file** - the project will be automatically loaded. For instance:
"C:\Program Files\DeskProto 7.0\deskproto.exe" C:\torso.dpj
This project may also be a Template project.

DPW
This is a **DeskProto Custom wizard file** - the custom wizard will be automatically started. For instance:
"C:\Program Files\DeskProto 7.0\deskproto.exe" "C:\Program Files\DeskProto 7.0\Wizard\123waxRing.dpw"

DPS
This is a **DeskProto Script file** - the script will be automatically started. For instance:
"C:\Program Files\DeskProto 7.0\deskproto.exe" "C:\Program Files\DeskProto 7.0\Scripts\ScriptedBottle.dps"

DXF, AI, EPS
These are CAD-files containing **Vector data** - the vector file will be automatically loaded. For instance
"C:\Program Files\DeskProto 7.0\deskproto.exe" "C:\ProgramData\DeskProto 7.0\Samples\DpBeerTray.dxf"
The default project needs to be a Vector project.
Note that DXF can contain Vector data and/or Geometry data. The default project type decides how the file is loaded.

STL, DXF
These are CAD-files containing **Geometry data** - the geometry file will be automatically loaded. For instance
"C:\Program Files\DeskProto 7.0\deskproto.exe" "C:\User\STLdata\medaillon.stl"
The default project needs to be a Geometry project.

BMP, GIF, JPG, JPEG, PNG, TIF, TIFF
These are CAD-files containing **Bitmap data** - the bitmap file will be automatically loaded. For instance
"C:\Program Files\DeskProto 7.0\deskproto.exe" "C:\ProgramData\DeskProto 7.0\Samples\XYZlogo.png"
The default project needs to be a Bitmap project.

You can test command line parameters by entering these command lines in the Search dialog of the Windows Start button.
Or you can edit the Shortcut on the desktop (click it with the right mouse-button, in the context menu choose Properties, select Tab Shortcut): you can add the command line parameter to field Target.
Or you can use the Windows Command window (start by entering CMD in the Search dialog) and enter the command in MS-Dos.

2. Second parameter:

The above options start DeskProto and load data, nothing more (except of course in scripts). Calculating toolpaths and writing the NC file still has to be done by the user. DeskProto allows you to automate this as well. You then need to start DeskProto with TWO command line parameters, the second being the name of the **NC file**.

For instance: issue the following command:

"C:\Program Files\DeskProto 7.0\deskproto.exe" C:\casting.stl C:\casting.nc

As a result DeskProto will be started, the file casting.stl will be loaded, toolpaths will be calculated, the NC file casting.nc will be written, and DeskProto will be shut down automatically. The user just sees DeskProto coming up and closing down again. The file extension of the NC-file needs not be the one that is prescribed by the postprocessor: the name on the command line will overwrite that.

The toolpath calculations are done using the parameters as set in the defaults. This works great when the application involves repeating the same job with a slightly different geometry. This is for instance used for making custom insoles: each sole is different, however all soles can be machined using exactly the same parameters.

An application programmer then can include DeskProto in his CAD application without bothering the end-user. Calling DeskProto with the correct command line parameters can be done by a macro that is called by a button in the CAD application.

Extra parameter:

One extra parameter is supported, which is not a file specification: adding "**/nocalculationwarnings**" as command line parameter will make DeskProto suppress calculation warnings. This flag may be added on any position on the command line, and does not count as first or second parameter.

This option is useful in case of automated processes, where it should not be needed for the user to give input (press OK) on warnings that are normal for that process. Only non-critical warnings are suppressed, for instance "*The selected curves of the operation go 5.00 mm outside of the material block*."

The first command line parameter may also be a DPJ file. When a DPJ file is loaded the parameters in this file will of course be used, not the current DeskProto defaults. This can be used when the CAD-file used has the same name for all models.

This option allows you to achieve full automation like mentioned above, for two sided milling.

For instance:

"C:\Program Files\DeskProto 7.0\deskproto.exe" C:\TwoSided.dpj C:\TwoSided.nc

When you do this with a template project file DeskProto will prompt the user to browse an STL file.
The project may not be a wizard-project, as then on opening the project the Wizard will be started which will stop the automation. So when you created the project using a wizard make sure to make (at least) one small change in the dialog-based interface before saving the DPJ file.

We strongly advise to define both command line parameters including a path specification. In case the first parameters comes with a path and the second does not, the NC file will be written in the directory of the first (STL or DPJ) file, as then that has become the "current folder".

The second command line parameter also works in case the pproject cause DeskProto to save more than one NC program file (for instance because of a toolchange): the standard DeskProto naming conventions will be used for the NC files.
In case the DPJ file that is loaded contains more than one part, then for all parts NC files will be written. Extra file names will be generated by extending the NC filename using the standard DeskProto naming conventions for multiple NC files.

5.24 Template projects

Some applications concern DeskProto projects that all very much look like each other: all settings equal, only the CAD-data are (slightly) different. In that case you don't want to again and again set all parameters for each next project.

A first possible solution is to then enter all parameters as **Defaults** (Default Part and Default Operation). When you then start DeskProto and load the CAD-data (vector, geometry or bitmap) for the new project, all parameters have already been set and you can immediately calculate the toolpaths. This even makes it possible to fully automate DeskProto: see the paragraph on Command Line Parameters.

Examples where the defaults can be successfully used:
- production of custom insoles: all products are machined with exactly the same settings, only the Geometry file is different.
- production of name-tags: again the setting are equal for all tags, only the Vector file is different for each product.

The above solution is useful only if **all** your projects need these same parameters. In case you have several types of projects, then the alternative solution is to create a project-file without CAD-data, which is called a **Template project**. In order to create such template project you will still need to load CAD data (vector, geometry or bitmap), as without CAD-data it will not be possible to set any Part parameters. When you have set all parameters that you need you can remove your CAD-data file (in the Project parameters), and then save the project.

DeskProto will warn you that no Vector-data / Geometry-data / Bitmap-data is present, and ask you if you want to create a template project. When you later open such template project, DeskProto will immediately show a File-Open dialog to load a CAD file into the project.

In DeskProto the first operation of a part defines the type of the project: when the first operation is a Vector operation the project is a Vector project. Same for template projects, which means that in these three cases DeskProto will treat a project that it opens as template project:
- first operation is a Vector operation, however no Vector-file is present
- first operation is a Geometry operation, however no Geometry-file is present

- first operation is a Bitmap operation, however no Bitmap-file is present

Here some examples where template projects may be useful:
- You might want to define a project-template you can use for a certain material. For instance when you always machine dental models your settings are always the same. However some models are in zirconium and some in wax, needing different settings.
- You might want to define a project-template for molds, using inverse milling.
- You might want to define a project-template with one part defined as top part, another defined as bottom part (with correct transformations), each containing one operation for roughing, and one for finishing.

After opening a template project and loading the CAD file that you want to use you need to take care NOT to save the project, as that will over-write your template project. If needed to save use Save_As, and a different name.

A second way of using template project files is to always use the same name for the CAD file. Your template project than will open this file when starting DeskProto: you just need to make sure that you have saved the CAD-data to be used in that CAD file before starting DeskProto.

Finally: the third option for automation that DeskProto offers is writing a Script.

Index

W

Z