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 8.0

Delft Spline Systems
Utrecht, The Netherlands
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 or 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 **Clipper** library for clipping and offsetting lines and polygons
Copyright © 1995-2013, Angus Johnson
Used and distributed under the Boost Software License V1.0.
angusj.com

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/zlib-ng/minizip-ng

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 **SVG++** library of functions to work with SVG files
Copyright © SVGPP.org (Oleg Maximenko)
Used and distributed under the Boost Software License V1.0.
svgpp.org

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

1.3 Essentials

What does DeskProto offer
DeskProto is a **CAM** program (**C**omputer **A**ided **M**anufacturing) for 3-axis, 4-axis and 5-axis CNC milling machines, offering **Desk**top **Proto**typing. DeskProto allows you to to create CNC toolpaths for 2D vector drawings, for 3D geometries, as well as for 3D reliefs based on digital images. 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 only 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, EPS or SVG file.
Geometry-data: 3D geometry defined as a collection of triangles (facets) that describe its outer surface (polygon data), stored as STL, 3MF or DXF file.
Bitmap-data: 2D image containing colored pixels, stored as BMP, JPG, GIF, PNG or TIF file.
So in fact DeskProto offers **three CAM programs for the price of one** !

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

What hardware/software is needed
DeskProto is available for Microsoft Windows, for Apple MacOS and for Linux. DeskProto is a 64 bits application.

For **Windows** it needs Win10 or newer.
Minimum required hardware is a PC with 4 GB RAM and 100 GB free disk space: faster/more is better. The graphics card needs to support OpenGL V2.1 or newer. A special build for Win XP (SP3, 32 bits) is available on request. ???? Win 7 ??

For **MacOS** it needs Mojave (10.14) or newer.

Here as well minimum hardware requirement is 4 GB RAM and 100 GB free disk space: faster/more is better.
The MacOS version does not include the "*Custom Wizards*" nor the option "*Send Toolpaths directly to machine*" (each of these a rarely used feature). As Apple's implementation of OpenGL is incomplete the commands "*Print Image*" (File menu) and "*Generate Contour*" (in the dialog to graphically set the Operation Area) do not work in the MacOS version,

For **Linux** it has been developed and tested using Ubuntu 20.04 (64 bits), still it should also work on a recent version of most other popular Linux distributions.
Here as well minimum hardware requirement is 4 GB RAM and 100 GB free disk space: faster/more is better.
The Linux version does not include the "*Custom Wizards*" nor the option "*Send Toolpaths directly to machine*" (each of these a rarely used feature).

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

1.4 Specifications

Required operating system (Microsoft Windows):
Windows 10 or newer (64 bits).
A special build for Win XP (SP3, 32 bits) is available on request. ????

Required operating system (Apple MacOS)
MacOS Mojave (10.14) or newer (64 bits).

Required operating system (Linux)
Ubuntu 20.04 or newer (64 bits). It should also work on most other popular Linux distributions.

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 4GB

Required graphics:
A 3D graphics card (or on-board graphics) that supports OpenGL V2.1 or newer,
The screen resolution of the computer needs to be at least 1024x768 pixels.

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 / 7.1
DPJ, version 8.0
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
SVG Scalable Vector Graphics
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
3MF **3**D Manufacturing Format
limited to only the 3D Model information
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, format and file extension are different per machine.

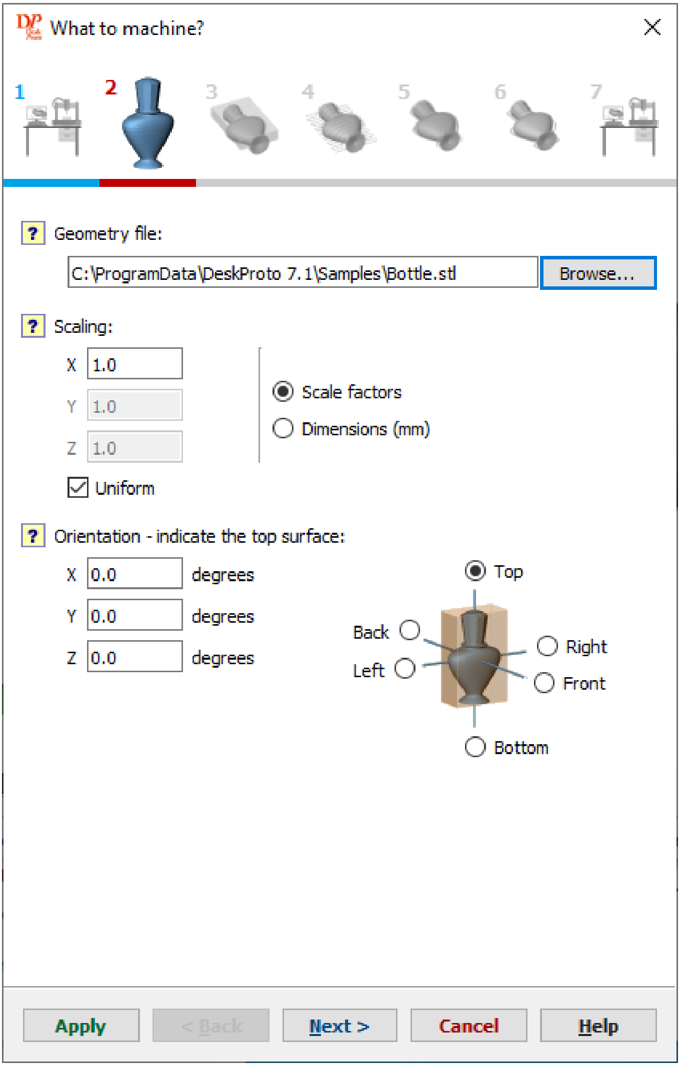
Other supported file-types:
DPW DeskProto Wizard
defines one custom wizard
DPS DeskProto Script
contains a DeskProto script.
DPT DeskProto Toolpath
contains all toolpaths for one DPJ file

Maximum File size
The DeskProto code does not have a maximum for any file. In practice the maximum is limited by the computer's resources. DeskProto is a 64 bits application, so the maximum STL file-size mainly depends on how much RAM memory is present: when the Operating System 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 even 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 wizards 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 all Wizards that are available, after selecting one and pressing next the actual wizard is started.



The illustration above shows a typical wizard screen.
On top six icons that indicate the sequence of pages in this wizard. These icons function as tabs: when colored they 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 the number of 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 toolpaths for that tab page 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 to these 'normal' wizards a 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. More information via the link.

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. The forum page offers a special search option for the forum, as the website's search option (top right corner of each website page) does not cover the forum.

4. Carefully read the appropriate sections in the **Manuals** (printed or PDF download).

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

In case this does not work you can of course contact Delft Spline Systems.
When you send us a project please do so using the ZIP file that you can generate via Create Problem Report (Help menu). Sending only 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. For products containing freeform surfaces it is absolutely necessary to create physical models in order to really evaluate the design, so from the start SIPSURF has featured a module to easily calculate CNC toolpaths. At that time creating physical models to evaluate CAD data was called Rapid Prototyping.

Since that time Rapid Prototyping has been recognized as vital for product development, a few years it has even been a buzz word. Based on a long experience of Rapid Prototyping and NC milling, Delft Spline Systems has been able to develop the 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 phrased in our motto

"CNC machining for non-machinists"

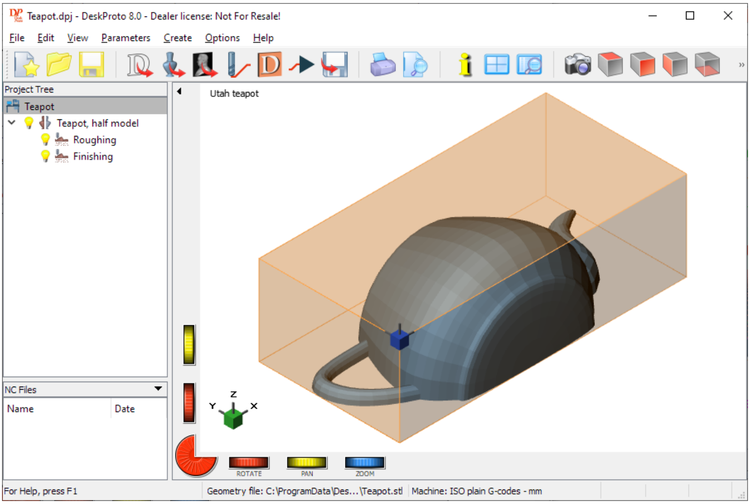
We hope you will enjoy using DeskProto, and we expect it will help you in producing high quality parts within a short time.

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

Delft Spline Systems
Utrecht, The Netherlands

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

II Screen layout



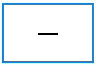
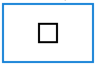
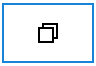
The computer screen presented by DeskProto looks like the screenshot shown above. In this chapter a short explanation will be given of each element on the screen. The image shows a Windows screen. Using MacOS or Linux the screen looks a bit different, still the same elements are present and can clearly be recognized. From top to bottom, the following elements are present:

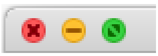
- The top line is called the **Title bar**. It contains the name of the current project, the word DeskProto, information about your license, and a few operating system icons.
- The **Menu bar** is the next horizontal line. It contains in black characters the names of the available pull-down menus. In MacOS the menu bar has a different location, and offers an extra menu (left of File) called DeskProto. That will all look familiar for Apple users.
- The button bar or **Toolbar** is the horizontal line below, containing a number of buttons for commands that need to be at hand. DeskProto offers these buttons in **five sizes** (size to be set in the View menu), the illustration shows Medium size buttons.
- The **View window**. The large screen-area below the button bar (on the right side 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.

This division of the main screen into three tiles (View, Project tree and NC files) can be changed. You can do so exactly as for the complete window: move your cursor over the border between two tiles (called the **splitter)**, see the cursor change, press the left mouse button and move. The NC files window and the Project tree window also can be completely made invisible using the small arrow buttons (the black triangles next to the splitters).

Just as for most dialog screens, in **Windows** you can **Resize** the complete DeskProto screen using the standard icons on the right end of the title bar. Two standard sizes and Custom are available (the exact shape and color of these buttons differs per Windows version):

-  The minimize button on the title bar will minimize: no window visible, only a button on the Taskbar.
-  The maximize button on the title bar will maximize: full screen window.
-  The custom size button restores the latest custom size. 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.

In **MacOS** you can use the  three colored buttons on the left side of the title bar (red, yellow and green), for closing, minimizing, and zooming (expanding to full-screen) the window. After zooming to full screen the title bar with the colored buttons will have disappeared, you can again see it by moving your cursor to the upper border of the screen.

In **Linux** the three buttons on the title bar look like this: . Functionality is the same as in Windows.

2.1 Title Bar

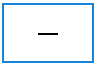
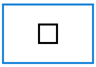
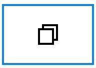
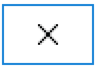
The Title bar is located along the top of a window.
It's exact appearance depends on your version of Linux/MacOS/Windows, and on the Theme that you have selected. In Windows this bar may be semi-transparent, which is called the Aero interface.

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

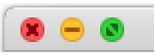
A title bar may contain the following elements:

- 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 8.0")
- License type: commercial, educational, hobby, etc. (here "Dealer license: Not For Resale").

In **Windows** these buttons are present (the exact shape and color of these buttons differs per Windows version):

- Application Control-menu button (the orange icon "DP")
- 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.

In **MacOS** the buttons are different:

-  These three buttons are for closing the dialog (red), minimizing it (yellow), and zooming (expanding to full-screen mode, green). The icons on these buttons are visible only when the cursor moves over them.

In **Linux** the buttons on the Title bar look like this:

-  Functionality of these three buttons is the same as in Windows, see above.

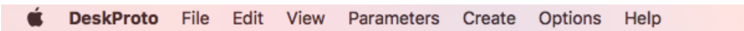
The Windows images were made in Win10, the MacOS buttons in Catalina, the Linux buttons in Ubuntu 18.04

2.2 Menu Bar

The menu-bar is the main route to all the commands that are available in DeskProto. This is why both the Help file and the Reference manual are 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 menu shown above is present both in **Windows** and in **Linux**. 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. This is a feature in Windows: 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 versions of Windows the underlining will be invisible: you can press the Alt button to make it visible.

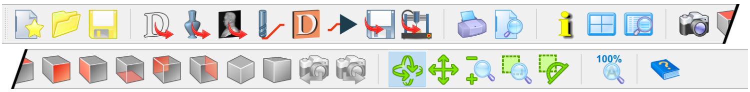


The menu bar in **MacOS** (Apple) contains one extra menu: the DeskProto menu. This is the first menu, before File: in MacOS for each application the first menu has the name of the application.

DeskProto	*About* / *Activate* / *Upgrade* / *Check f Upd* / *Preferences*

The commands in this menu are present in Windows as well: in the Help menu and the Options menu.

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 "Toolbar Button Size", with which you can choose between five sizes of buttons. The best choice will depend on the resolution of your screen: on a high-res screen you may want larger buttons, as the button sizes are in pixels. The size of the buttons can also be changed in the Preferences: on the Advanced tab you can set a scaling factor for the complete user-interface.

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.

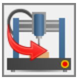
Load a Bitmap file into the project (adding is not possible: max 1 bitmap file loaded).

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), showing its current state.

Show an Animation of the toolpaths: your will see a cutter move over your part, following and drawing the toolpath. This is a toggle button as well.

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 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 visible 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).

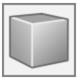
Set the viewpoint of the active view to Left side view (-90 / 90 / 0).

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 Right 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 action).

Change mouse-function to rotation.
The four mouse-buttons are "toggle buttons", showing depressed in case selected. De-selecting can be done only 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.

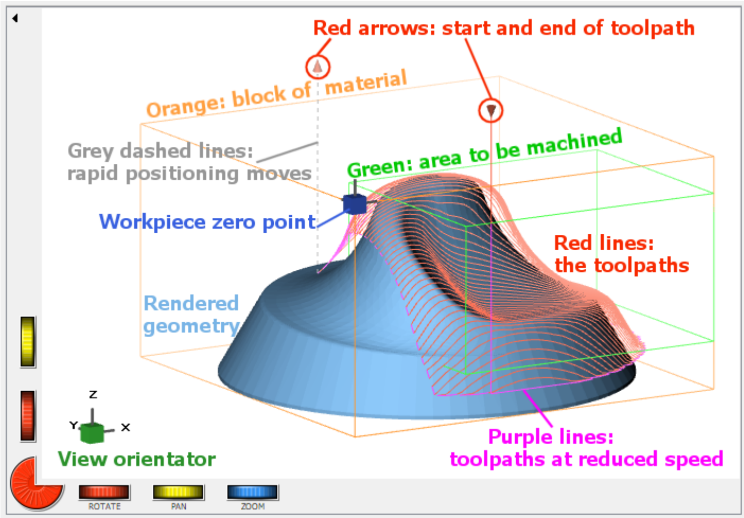
Change mouse-function to measuring: click two points on the screen to find the distance between (shown on the status bar).

Note that one of these five 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 the view, and moving the mouse with the wheel pressed pans the view.

Set the Zoom percentage to 100% to show the complete material block.

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. Note that this screen is about geometry machining: for vector projects and bitmap projects other items can be visible.

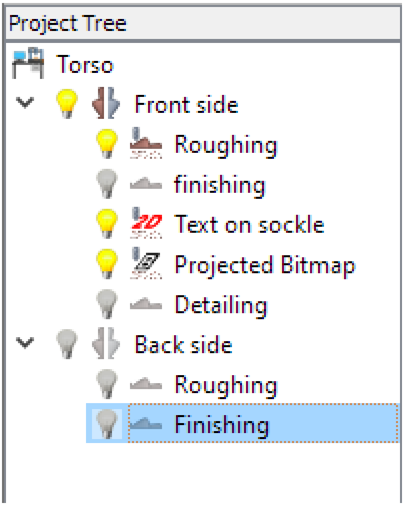
The wireframe block in orange is the **Material block** that you start with for this Part.
The wireframe block in green is the **Area to be machined** for one Operation
Both blocks can be shown shaded by making them Translucent in the **Items Visible** dialog
Drawn in metallic blue is the **Geometry** 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 covers the most used items in the View Window. Many more items can be displayed though: these 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. For Operation lamp icons two keyboard/mouse shortcuts are present: Ctrl+Click means "Make Visible and Hide Others", Ctrl+Shift+Click means "Make All (in)visible" (on a Mac use the Command key instead of Ctrl).

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 get a blue background 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:

Open the parameters dialog of that tree-item.

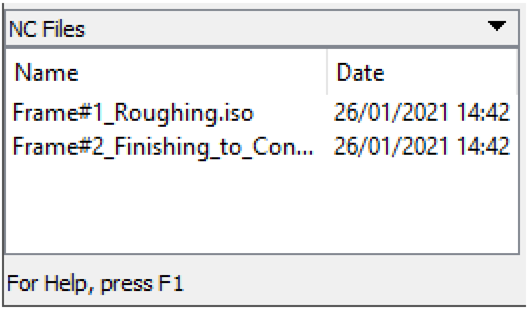
Add a Part to the project. The settings of the default part are used.
Add an Operation to that part. The settings of the first default operation are used. There are Add options: for a Vector Operation, a Geometry Operation and a Bitmap Operation.
Copy an operation or a part will add an item that is identical 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). Note that moving several parts is easier in the Project parameters dialog, and moving several operations is easier in the Part parameters dialog.

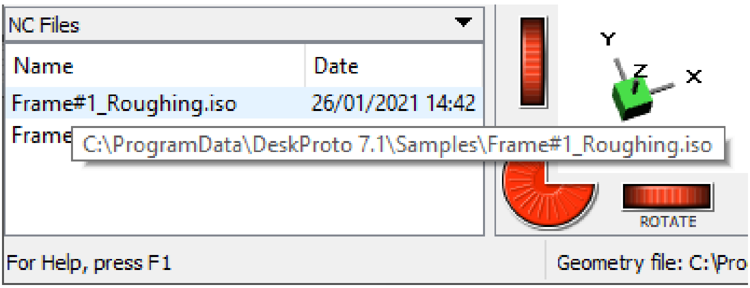
The other options in the context menu will be self explaining.

2.6 NC Files



The NC File List 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.

The NC Files window can made visible or invisible by checking or unchecking the option NC File 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.



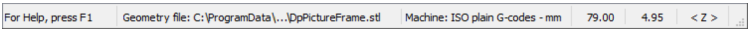
Hovering over the filename (positioning the cursor there without clicking) will make DeskProto show the complete file specification of the NC file in a tooltip: see the screenshot above.

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 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.

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 shows the name of each menu command and each toolbar button when the cursor is hovers over that command or button (a sort of extra Help information). Menu commands are also described when you use the arrow keys to navigate through the menus.
The left side of the status bar is also used to display the results of the measuring tool.

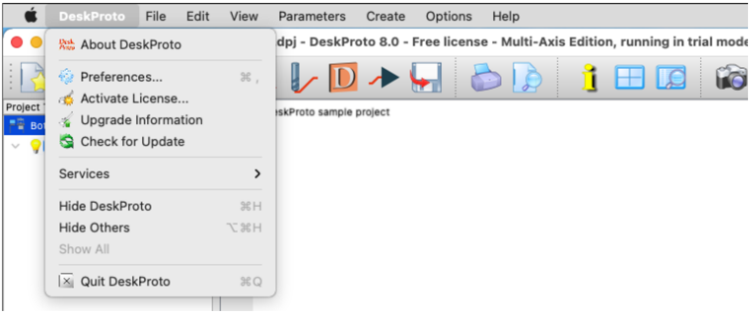
In the middle area the most important parameters of the selected tree-item are displayed: CAD filename and machine for the Project, dimensions for a Part, cutter and precision for an Operation.

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 six 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.

For advanced users: when mouse function "Measuring" is active the coordinates that are shown can be changed by pressing a keyboard key while moving the cursor:
Alt: the coordinates are shown without the translation that is set in the Part parameters.
Shift: the coordinates are shown in the alternate unit system, so in inches when you work in mm, in mm when you work in inches.

III Menu commands

3.1 DeskProto Menu (MacOS only)

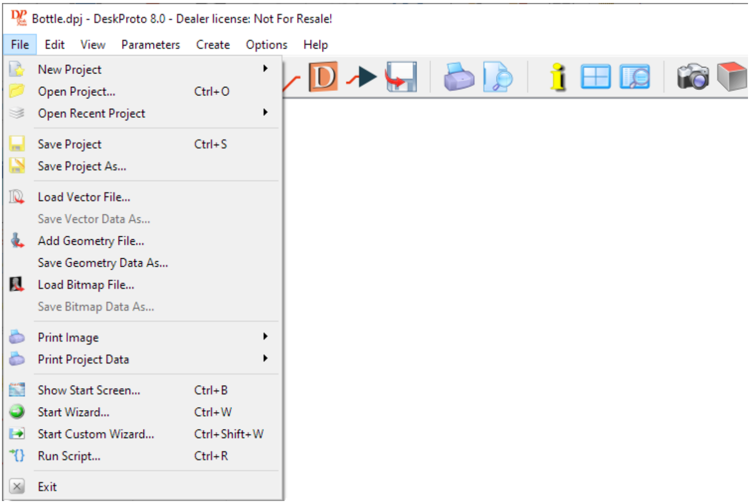


In every MacOS application the first menu has the same name as the application, containing some standard options for the application. The image above shows the DeskProto menu.
In Windows and in Linux these commands are all present as well, however there you can find them in other menus.

In this document the commands are listed conform the menu structure in Linux and Windows,
MacOS users can find more information about these six DeskProto commands via the links below:
About DeskProto
Preferences
Activate License
Upgrade information
Check for Update
Quit DeskProto

Services, Hide and Show are standard MacOS options, not related to DeskProto

3.2 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+O" means to keep the Ctrl key pressed and then press the O key: DeskProto will perform the Open Project command (on a Mac use the Command key instead of Ctrl). No need to use your mouse !

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

3.2.1 New Project

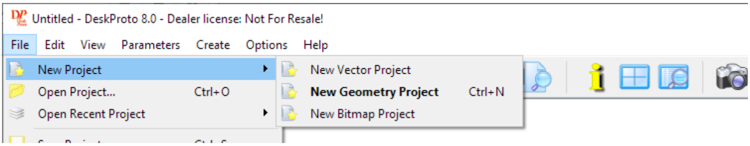
This File menu command creates a new project in DeskProto.
This project will be called "Untitled" until it has been saved.
You can select one of three project types: Vector project, Geometry project, Bitmap project,
by selecting one in a sub-menu.

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 (like geometry **and** vector) 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 (Windows, Linux) ⌘+N (MacOS).

3.2.2 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 (Windows, Linux) ⌘+O (MacOS).

When opening a project file, the CAD file(s) that were used will be opened and loaded again: the DPJ file 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 someone 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.

3.2.3 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**. Alternative name is MRU, which stands for Most Recently Used.
It is a convenient standard option in current software.

3.2.4 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 the Save As command.

Note that the project file does not contain the CAD-data, only links to the CAD-files. When you need to save the geometry after it has been changed use the Save Geometry As command. Similar for Save Vector Data As and Save Bitmap Data as.

Shortcuts:

Toolbar: 

Keys: Ctrl+S (Windows, Linux) ⌘+S (MacOS).

3.2.5 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).

When you have loaded only one CAD file DeskProto will suggest to use the name of that file for the project to be saved.

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

3.2.6 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, SVG, AI and EPS (AI en EPS need to be version Adobe Ilustrator 8 files).
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. For an overview of all Vector files that you loaded, or when you want to replace your current Vector file with a new, go to the Parameters menu, choose the option Project Parameters and go to tab Vector.

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 CAD data.

3.2.7 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 in a CAD file with one or more Transformations applied
2- merge several vector files to one combined new CAD file.
3- save vector data in a different format (so use DeskProto as a converter for vector drawing 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 four file formats: DXF, EPS, AI and SVG.
However, for exporting Vector files only DXF and SVG are supported.

3.2.8 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, 3MF, 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. For an overview of all Geometry files that you loaded, or when 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.

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 CAD data.

3.2.9 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
3MF	3D Manufacturing Format	
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 widely used, and the binary version results in a much smaller file size than ASCII. So Binary STL is the default file type here

3.2.10 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. In DeskProto the image will be positioned at the top of the block, with its bottom left corner at point X=0 Y=0 in CAD coordinates.

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

3.2.11 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.

3.2.12 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 (Windows, Linux) ⌘+P (MacOS).

3.2.12.1 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 as it will be printed. 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.

On most MacOS versions this command cannot be used as Apple does not offer the full OpenGL functionality.

Shortcuts:

Toolbar:  (for Print Image Preview)

3.2.12.2 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.

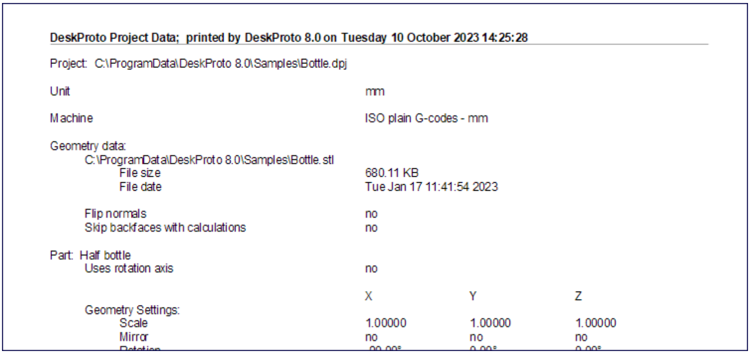
- Print Project Data Page Setup, for printing the Project Data.

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.

3.2.13 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 8.0 on Tuesday 10 October 2023 14:25:28

Project: C:\ProgramData\DeskProto 8.0\Samples\Bottle.dpj

Unit	mm		
Machine	ISO plain G-codes - mm		
Geometry data:			
C:\ProgramData\DeskProto 8.0\Samples\Bottle.stl			
File size	680.11 KB		
File date	Tue Jan 17 11:41:54 2023		
Flip normals	no		
Skip backfaces with calculations	no		
Part: Half bottle			
Uses rotation axis	no		
	X	Y	Z
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.

3.2.13.1 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 as it will be printed. 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.

On most MacOS versions this command cannot be used as Apple does not offer the full OpenGL functionality.

Shortcuts:

Toolbar:  (for Print Image Preview)

3.2.13.2 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.

3.2.14 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).

Shortcut:

Keys: Ctrl+B (Windows, Linux) ⌘+B (MacOS).
B stands for "Begin", as the S already is being used (for Save project).

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:

- Start a new project
- Open an existing project

- Other tasks

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

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.
When starting a new project using dialogs you need to select which project type. The difference between Vector/Geometry/Bitmap projects is explained on page New project.

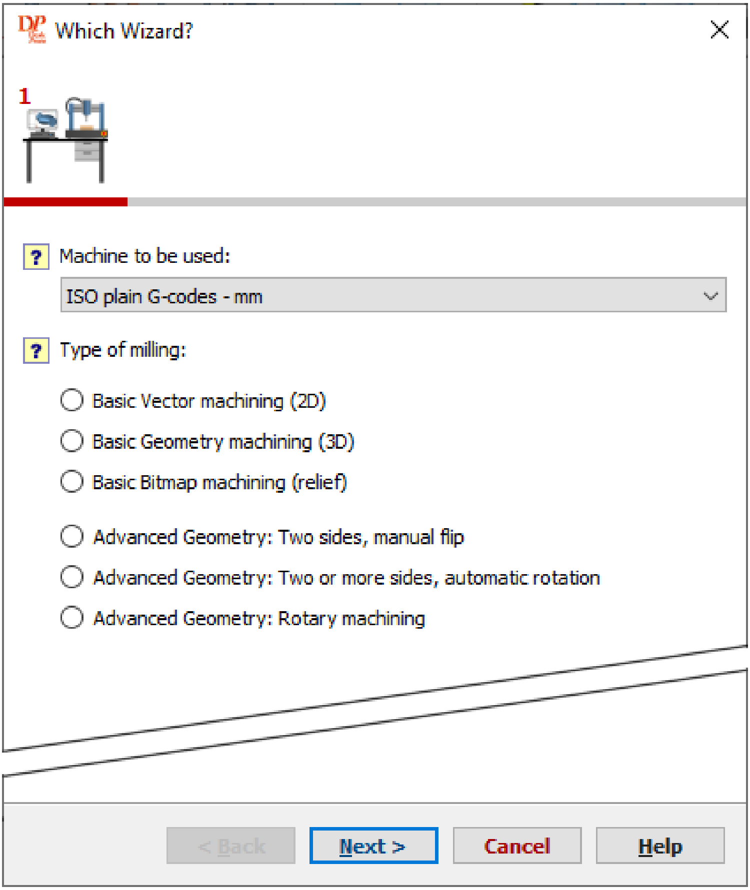
The DeskProto Setup has installed a number of **Sample projects** and **Sample geometries** on your PC. Including some great geometries like the DeskProto picture frame: see the lessons in the DeskProto Tutorial book.
Conform Microsoft's specifications the **Samples** have been installed in the \ProgramData\ 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 (no administrative rights) other locations are not permitted by Microsoft. Also when using MacOS or Linux the Samples folder may not be easy to find for you.
Using the checkbox **Use samples folder** will make DeskProto open the sample folder when you need to load the CAD data.

To open an **existing project** two options are present:
Command **Open project** will allow you to open any project file by browsing one.
Recent projects will be available only when you have used DeskProto V8 before. In this dialog only the latest 5 projects will be shown, more may be visible when you select this option via the File menu.

In the **Other tasks** section, **Open sample project** makes it easy to find the sample projects.
The other three 'other tasks' (Read tutorial book, View tutorial videos and Visit user forum) 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.

3.2.15 Start Wizard



The Wizard-based user interface is an important feature of DeskProto. It makes it possible for users without much CAM know-how to easily create the toolpaths that they need for their projects. Each wizard is a series of dialog screens (pages) that need to be filled in: the wizard so to speak *'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.

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 its own Help information page.

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

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

Calculate will calculate the toolpaths 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, SVG). It is meant for novice DeskProto users, and explains this procedure step-by-step. You can choose to use either Profiling, Pocketing or V-Carving toolpaths.

Basic Geometry Machining
This wizard creates 3D toolpaths for just one Geometry file (STL, 3MF, 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.

Two or more sides, automatic rotation, also called the **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 XYZ 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 all around: rotation axis machining. In contrast to the previous wizard, now the material rotates during machining (continuous rotation).
A rotation axis is an extra piece of equipment that lets your material rotate on the machine (just like a roast 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.

Shortcut:

Keys: Ctrl+W (Windows, Linux) ⌘+W (MacOS).

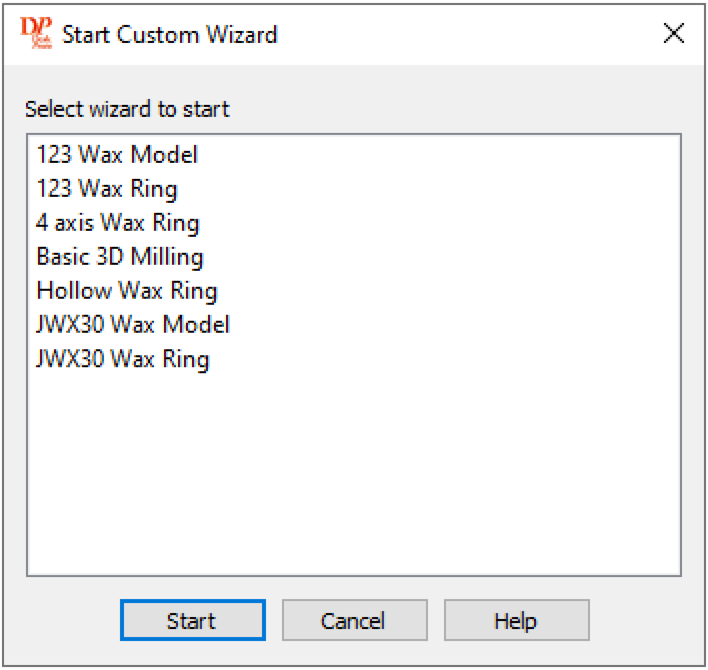
3.2.16 Start Custom Wizard

A Custom Wizard (also called Script wizard) is a wizard written in a Script language. It is stored as a separate file: a .DPW file (so it is not part of the file DeskProto.exe like the 'normal' wizards). This is a very powerful option, as any user (and any reseller) is able to create a custom wizard for a specific application and/or a specific fixture.

Custom wizards are not available in MacOS and Linux, only in Windows, sorry about that.

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, and contains all valid Wizards found in the subdirectory \CustomWizards\ of DeskProto's ProgramData folder, for instance in *C:\ProgramData\DeskProto 8.0\CustomWizards*
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 its 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 creating Custom wizards can be found on the Custom Wizard page.

3.2.17 Run Script

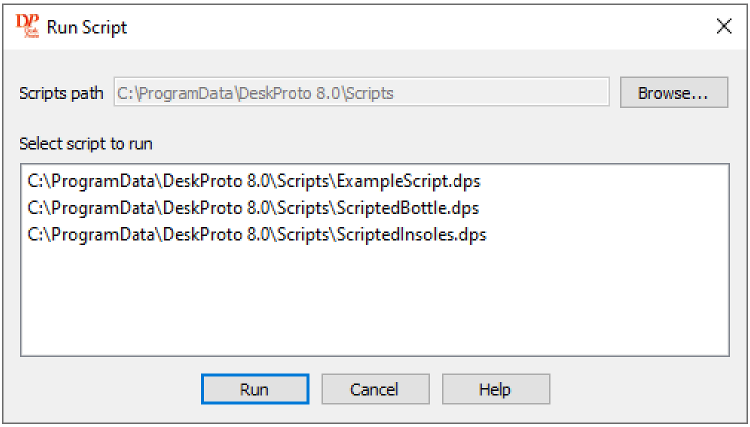
Normally when you use DeskProto you will load a CAD-file, set a number of parameters, calculate toolpaths and save an NC program file. So you manually perform a number of actions, in a specific sequence. In some applications the actions to be done and the parameter settings to be used are always the 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 (commands to be executed), 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.

Shortcut:

Keys: Ctrl+R (Windows, Linux) ⌘+R (MacOS).



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 (or Custom 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. Lines 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. Each script file should start with a few standards lines, which are nog displayed here.

```
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.

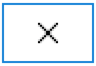
When using such script as command line parameter you can even add an Exit command: then the actual user does not need to perform any user-interaction with DeskProto.

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.

3.2.18 Exit

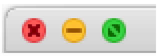
This command ends your DeskProto session. DeskProto will then prompt you to save projects with unsaved changes. In Windows and in Linux it can be found in the Help menu, in MacOS in the DeskProto menu (as "Quit DeskProto").

Windows shortcuts:

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

Keys: ALT+F4

MacOS shortcuts:

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

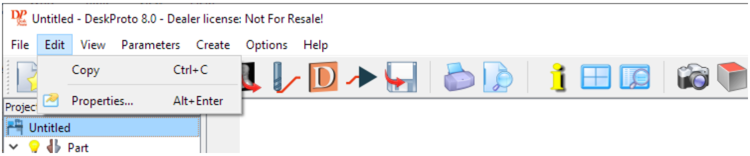
Keys: ⌘+Q

Linux shortcuts:

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

Keys: Ctrl+Q

3.3 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.

3.3.1 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 the Project line in the Tree is active the Copy command will make an error 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. So in this case the Copying is not to the clipboard: Copy will immediately add a line to the tree.

When the View Window is active the image in this window will be copied to the Windows Clipboard.
The clipboard is used to cut and paste data between windows applications: you can for instance later 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 (Windows, Linux) ⌘+C (MacOS).

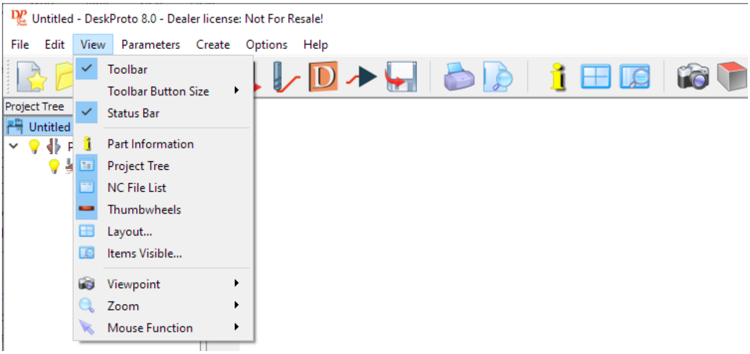
3.3.2 Properties

This Edit menu command can be used to edit the highlighted tree-item or the active view. Depending on what is selected 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 (Windows, Linux) Option+Enter (MacOS: this is what these weird icons mean).

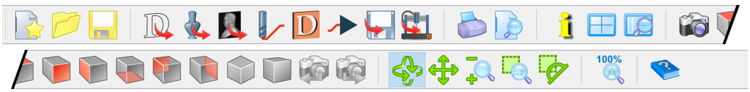
3.4 View Menu



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

In the screenshot above you can see that in front of each menu options an icon is present (or an empty space). For options that offer an on/off setting the background of the icon is blue (or blue with a "V" icon) when the option is on (so when that control is visible in the user interface), and the background is gray when the option if off.

3.4.1 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 "Toolbar Button Size", with which you can choose between five sizes of buttons. The best choice will depend on the resolution of your screen: on a high-res screen you may want larger buttons, as the button sizes are in pixels. The size of the buttons can also be changed in the Preferences: on the Advanced tab you can set a scaling factor for the complete user-interface.

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.

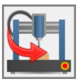
Load a Bitmap file into the project (adding is not possible: max 1 bitmap file loaded).

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), showing its current state.

Show an Animation of the toolpaths: your will see a cutter move over your part, following and drawing the toolpath. This is a toggle button as well.

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 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.

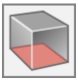
Change which Items should be visible in the active view (the Scene).

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 Left 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 Right 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 action).

Change mouse-function to rotation.
The four mouse-buttons are "toggle buttons", showing depressed in case selected. De-selecting can be done only 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.

Change mouse-function to measuring: click two points on the screen to find the distance between (shown on the status bar).

Note that one of these five 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 the view, and moving the mouse with the wheel pressed pans the view.

Set the Zoom percentage to 100% to show the complete material block.

Display the Help Topics dialog.

3.4.2 Toolbar Button Size

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. Or when you have a small monitor they might be to large as not all buttons will be visible: the toolbar is wider than your screen.

To solve that problem you can ask DeskProto to change the size of these buttons.
A black dot on a blue background will clearly indicate which buttons size currently is selected in the menu.

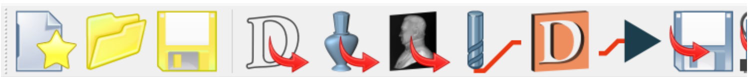
These are the five size options (the default size is Medium):

Extra small - 16 x 16 pixels

Small - 24 x 24 pixels

Medium - 32 x 32 pixels

Large - 48 x 48 pixels

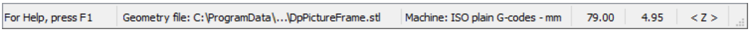


Extra large - 64 x 64 pixels

When you change the size of the toolbar buttons and also want to change the font size for all text (so resize the complete user interface - for all Windows programs) you can do that in the Windows Display properties (for MS Windows).

In addition to these five sizes for the buttons DeskProto offers an option to scale the complete user interface, on the Advanced tab of the Preferences (select "User Interface"). When you select a scaling factor that is larger than 1 the resolution of the toolbar icon images can be made even higher than 64x64 pixels (maximum 128x128).

3.4.3 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 shows the name of each menu command and each toolbar button when the cursor is hovers over that command or button (a sort of extra Help information). Menu commands are also described when you use the arrow keys to navigate through the menus.
The left side of the status bar is also used to display the results of the measuring tool.

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 six 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.

For advanced users: when mouse function "Measuring" is active the coordinates that are shown can be changed by pressing a keyboard key while moving the cursor:
Alt: the coordinates are shown without the translation that is set in the Part parameters.
Shift: the coordinates are shown in the alternate unit system, so in inches when you work in mm, in mm when you work in inches.

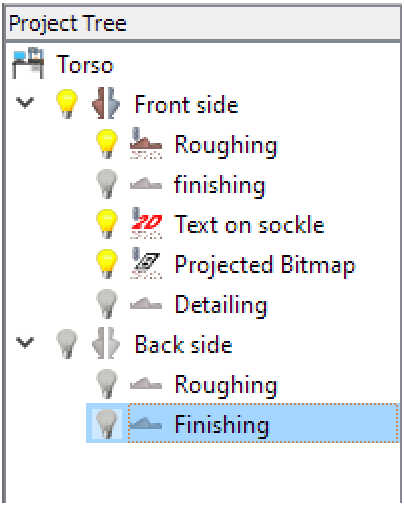
3.4.4 Part Information

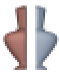
The View menu command Geometry Information displays or hides the **Part Information dialog**, which shows information about the geometry and the current part. A check mark is shown next to the menu item when the dialog is displayed.

Shortcuts

Toolbar: 

3.4.5 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. For Operation lamp icons two keyboard/mouse shortcuts are present: Ctrl+Click means "Make Visible and Hide Others", Ctrl+Shift+Click means "Make All (in)visible" (on a Mac use the Command key instead of Ctrl).

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 get a blue background 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:

Open the parameters dialog of that tree-item.

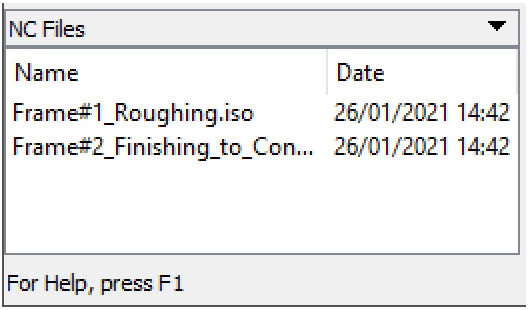
Add a Part to the project. The settings of the default part are used.
Add an Operation to that part. The settings of the first default operation are used. There are Add options: for a Vector Operation, a Geometry Operation and a Bitmap Operation.
Copy an operation or a part will add an item that is identical 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). Note that moving several parts is easier in the Project parameters dialog, and moving several operations is easier in the Part parameters dialog.

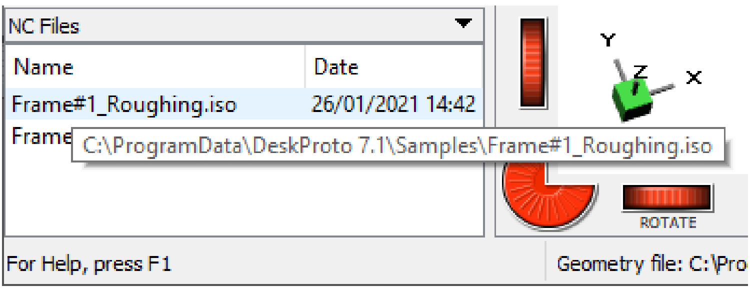
The other options in the context menu will be self explaining.

3.4.6 NC File List



The NC File List 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.

The NC Files window can made visible or invisible by checking or unchecking the option NC File 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.



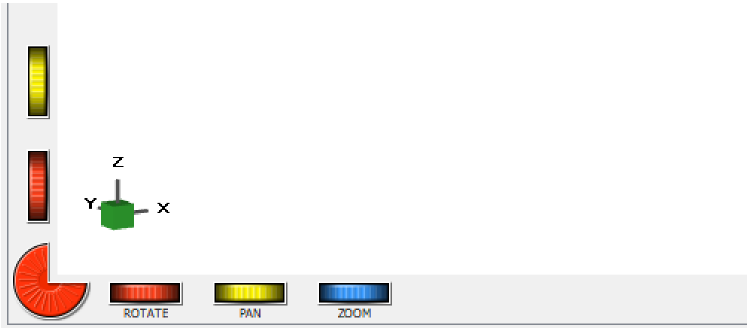
Hovering over the filename (positioning the cursor there without clicking) will make DeskProto show the complete file specification of the NC file in a tooltip: see the screenshot above.

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 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.

3.4.7 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.
Note that these rotations only change the viewing angle (camera position), not the orientation of the part in space.

A trick that may be handy: when you keep the shift key pressed while rotating the rotation will be done in steps of 15 degrees.

You can make the thumb-wheels visible and invisible using the Thumb-wheels command in the View menu.

3.4.8 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: 

3.4.9 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.

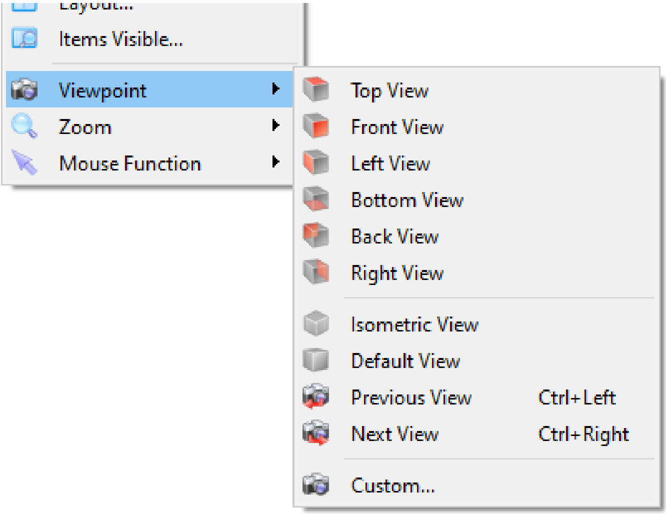
Shortcuts

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.

3.4.10 Viewpoint

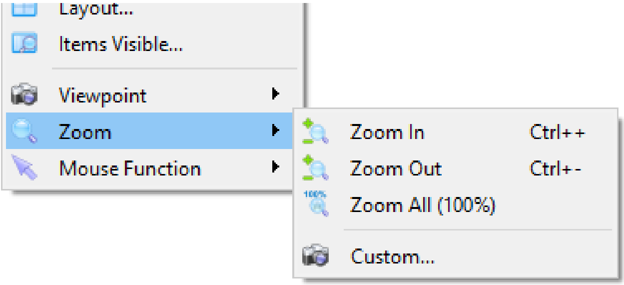


The View -> Viewpoint submenu offers the following commands to set the Viewpoint:

Top / Front / Left / Bottom / Back / Right 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.

3.4.11 Zoom

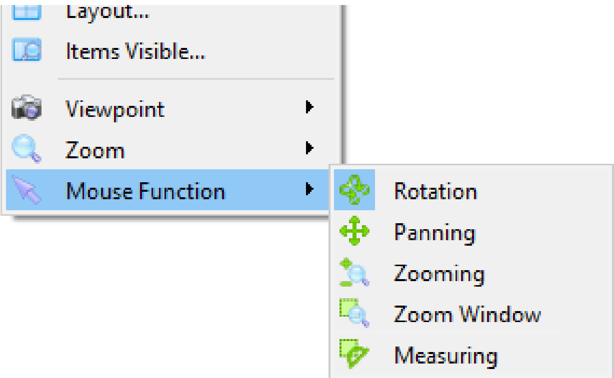


The View -> Zoom submenu offers the following commands:

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.

Note that the Zoom can also be set using the Thumb-wheels or the Mouse functions.

3.4.12 Mouse Function



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

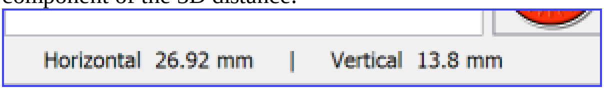
Rotation: use the mouse (move it inside the graphics view with the left mouse button pressed) to rotate your geometry. Imagine that the geometry is inside a large hollow glass sphere: with your mouse you can grab the sphere anywhere, and rotate it round its center point, including the geometry. This 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.
Some CAD programs (for instance Rhino) keep the Z-axis vertically aligned on the screen while rotating. When you prefer DeskProto to do that as well you can check "Lock Z-axis vertical on screen" in the Advanced Preferences for the Mouse.
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.

Panning: 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.

Zooming: 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. When you want these two actions reversed you can configure that in the Advanced Preferences for the Mouse. Note that the icon on the toolbar button then changes as well.

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.

Measuring: use the mouse to measure the distance between two points, by clicking these points on the screen. The result is **shown on the status bar** of DeskProto's main window. Note that the result is not always the real distance in 3D: only the horizontal distance and vertical distance as seen on the screen (so in a plane parallel to your screen) are measured. If one point is further away from the viewer then the other this tool does not include that component of the 3D distance.



How many digits are shown behind the decimal point depends on the size of the dimension: the smaller the more digits.

The measuring tool offers several extra options for the results to be displayed, by pressing a keyboard key while measuring:
Alt (for Apple the Option key): you get more detailed results, like "_ 53. 71 58. 07° \ 101. 5 31. 93° | 86. 17", in which
_ is the horizontal distance
angle of the diagonal with the horizontal line
\ is length of the diagonal
angle of the diagonal with the vertical line
| is the vertical distance
In this view the units are not specified.
Shift: the results are shown in the alternate unit system, so in inches when you work in mm, in mm when you work in inches.
Control (for Apple the Command key): the resulting dimensions are shown in pixels (more interesting for the developers than for users).
When you use the measuring tool in an empty project (no block dimensions, no CAD data) 1 pixel will be shown as 1 mm.

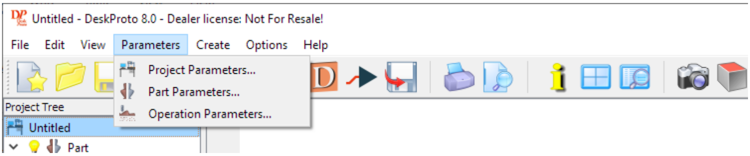
Using the first three mouse functions the geometry is continuously redrawn during the mouse movement. Depending on the size of your file and the speed of both your computer and graphics card, this redrawing will take more or less time. In case the redrawing is too slow, you can influence the number of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

An alternative for *Zoom* is rotating the **Mouse wheel**. This works no matter which of the above buttons is active. In the Advanced Preferences for the Mouse you can reverse the effect of the rotation direction ad also change the zooming speed when rotating the mouse wheel.
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.

The rotation tool and the panning tool offer an extra option: when you press the Shift button on your keyboard the two functionalities are switched. This is done for users who have a two-button mouse (so without a mouse wheel): when mouse function Rotation is active they can pan their view by pressing shift, so without having to select a different mouse-function.

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

3.5 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.

3.5.1 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: the first option.

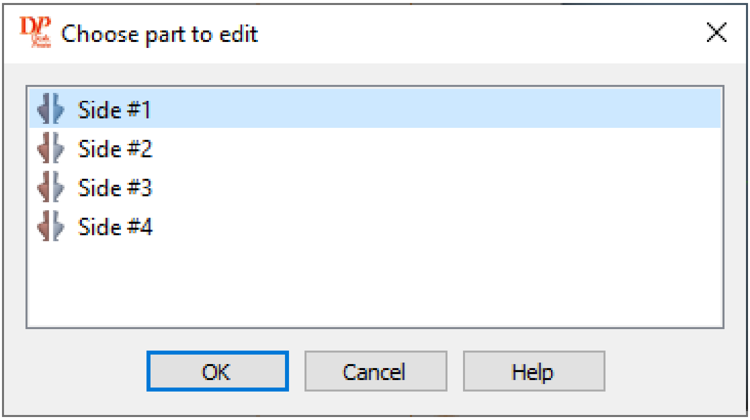
Shortcuts:
You can double-click on the project-line in the project tree (the base level item).
Or right-click on the project-line and select Project Parameters in the context-menu.

This same dialog is used for the Default Project parameters.

3.5.2 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.



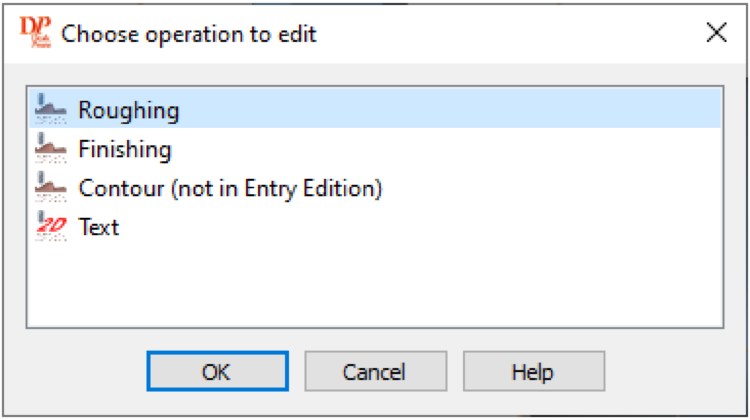
The part you select here will also become the current Part, and thus will be displayed when you have finished editing.

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

3.5.3 Operation Parameters

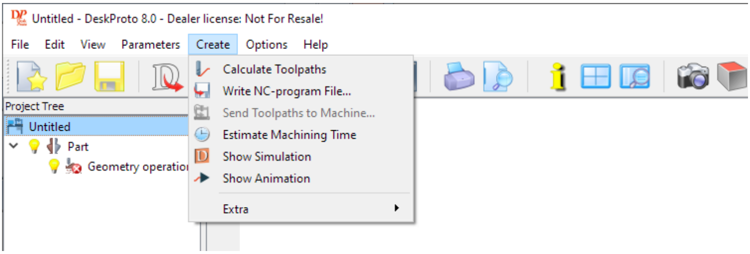
This Parameters menu command displays the Operation Parameters dialog in which you can edit the parameters of an Operation.
Operations can be either Geometry Operations , Vector Operations or Bitmap Operations.

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-line in the project tree (one of the third level items).
Or right-click on a operation-line and select Operation Parameters in the context-menu.

3.6 Create Menu

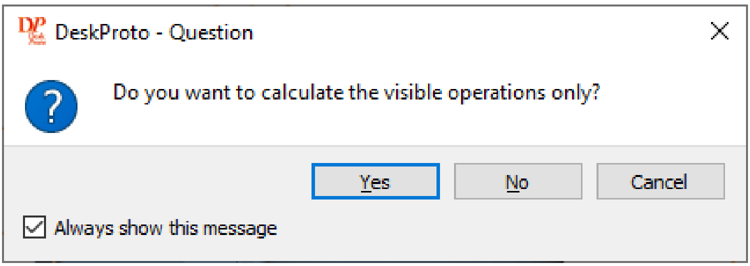


These options control all actions for calculating, evaluating and saving NC toolpaths. Note the submenu called Extra which offers some extra options. As these extra commands are very specific and not important for most users these commands have been "hidden" in a submenu.

3.6.1 Calculate Toolpaths

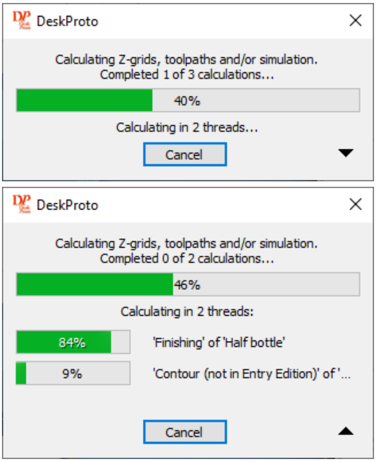
With this option you can create the toolpaths for the current part.

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 invisible, 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 in order to calculate 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.

Shortcut:

Toolbar: 

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

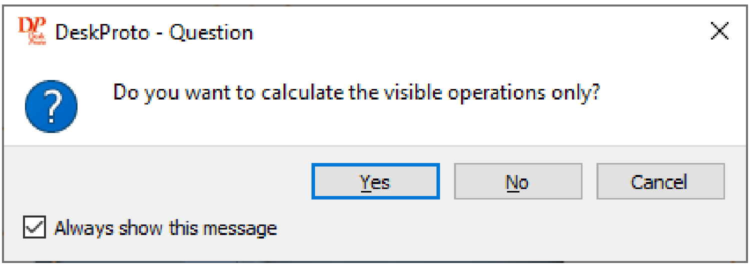
3.6.2 Write NC-program File

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.

DeskProto will ask you to give a **Name** for the NC-program file, in a standard Save-As dialog, in which the correct file-extension for your machine has been already entered. DeskProto will even suggest a name: the name of the project (only when the project already has been given a name). The only things you need to do are entering (or checking) the filename and making sure the file is being saved in the right place.

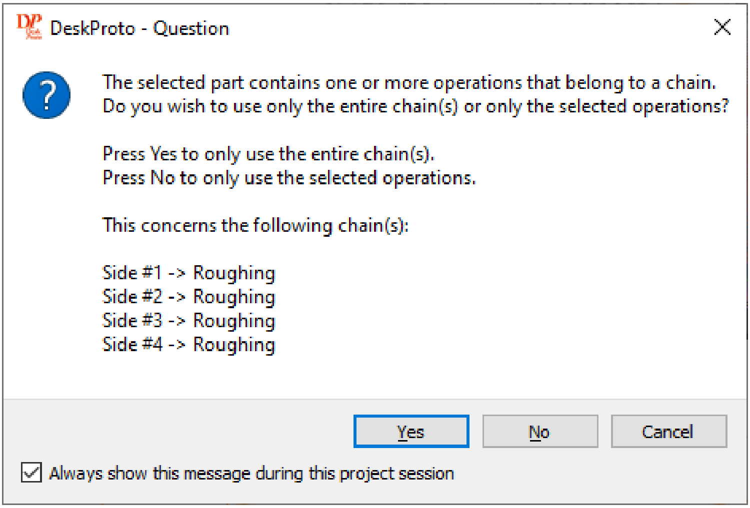
The name for the NC file that you enter here may be automatically changed. When you have more than one Operation and different cutters are used in these Operations, a tool change will be needed. For machines that do not support a tool change two or more NC files will be written (this behavior can be set in the postprocessor). 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
Note: Operation names may contain "reserved characters" that are not permitted in file-names (for instance * / : < >). DeskProto will replace reserved characters and spaces by underscores ("_").

In case all operations of the current part are visible (see Visible Operations), the NC file(s) will be written for all the toolpaths of the current part. 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 the NC file or if you want to use all operations.



In case you choose No (so choose ‘all operations’), they will also all be made visible afterwards.

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.

Other project warnings may be shows as well.

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: 

3.6.3 Send Toolpaths to Machine

For some CNC milling machines this option can be used to directly send the toolpaths to the machine (or to the machine's control software). 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.

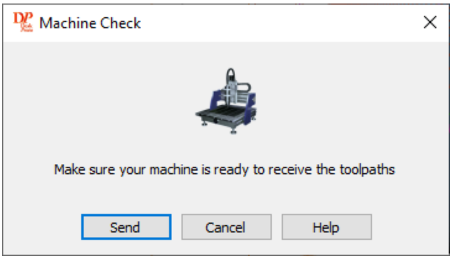
This feature is not available in MacOS and Linux, only in Windows, sorry about that.

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. In fact you can select any program, so (for instance) a milling simulation software is possible as well, or (for the diehards) a plain text editor like Notepad++ to edit 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 existing 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 load the NC file in the machine's control software (or transfer the NC file to the controller using the machine’s 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 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. Open the NC-program file to the control software.
4. 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 tab 'Zero-point' of the Part Parameters.
5. Start the machining process.

3.6.4 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. The help information for the dialog that was just mentioned explains why.

3.6.5 Show Simulation

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

When the simulation is shown DeskProto will automatically make many other items on screen invisible: Material block, Operation areas, toolpaths, Z-grids. You can override that via the Items Visible dialog.

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

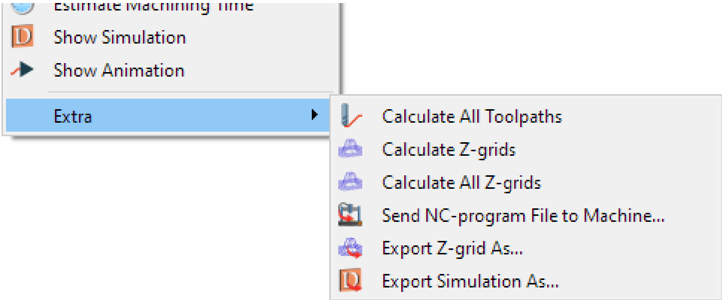
3.6.6 Show Animation

With this option you can show an Animation of the toolpaths for the current part.

In the Toolpath Animation dialog that pops up you can control the animation display.

When an animation is shown DeskProto will automatically make some other items on screen invisible: geometry, toolpaths, Z-grids, simulation. You can override that via the Items Visible dialog.

3.6.7 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.

3.6.7.1 Calculate All Toolpaths

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

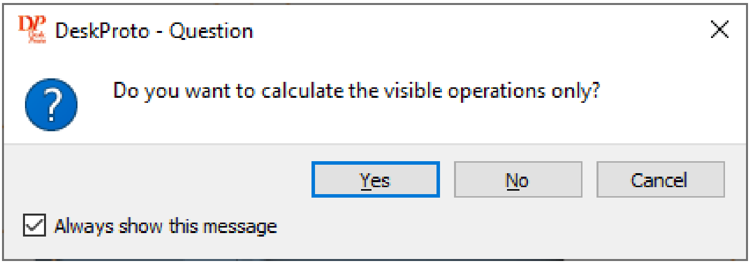
Where the other commands in the Create menu only concern the current part, this one command influences all parts in the current project.

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

3.6.7.2 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.

3.6.7.3 Calculate All Z-Grids

With this option you can create all Z-Grids 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).

3.6.7.4 Send NC-program File to Machine

For some 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.

This feature is not available in MacOS and Linux, only in Windows, sorry about that.

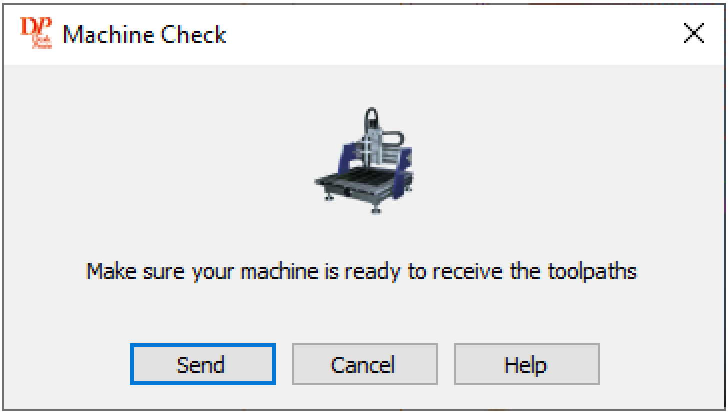
In the Preferences (Options menu) the following destinations can be configured for this option:

- 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 edit 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 load the NC file in the machine's control software (or transfer the NC file to the controller using the machine’s 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.

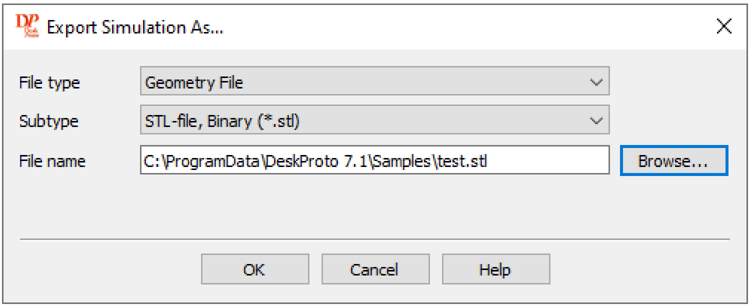
How to prepare the machine will be different for every machine. Still some general guidelines can be given. Basically the following steps have to be taken:

1. Fixture a 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. Open the NC-program file to the control software.
4. 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 tab 'Zero-point' of the Part Parameters.
5. Start the machining process.

3.6.7.5 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 Simulation. For exporting the Z-grid 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 has been described above.

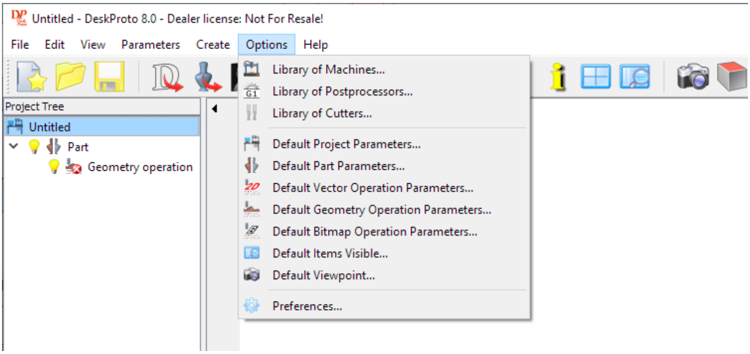
For export as Bitmap file DeskProto will convert the 3D information to 2D by converting Z-height to gray-value. For each point in the Z-grid one pixel will be made. The highest Z-value will be given a white pixel, the lowest level a black pixel, and all in-between values an according in-between gray 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, 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, the three coordinates of one point per line.

The function of **File name** will be clear, you need to use the Browse to fill this field or edit its contents (both file name and location).

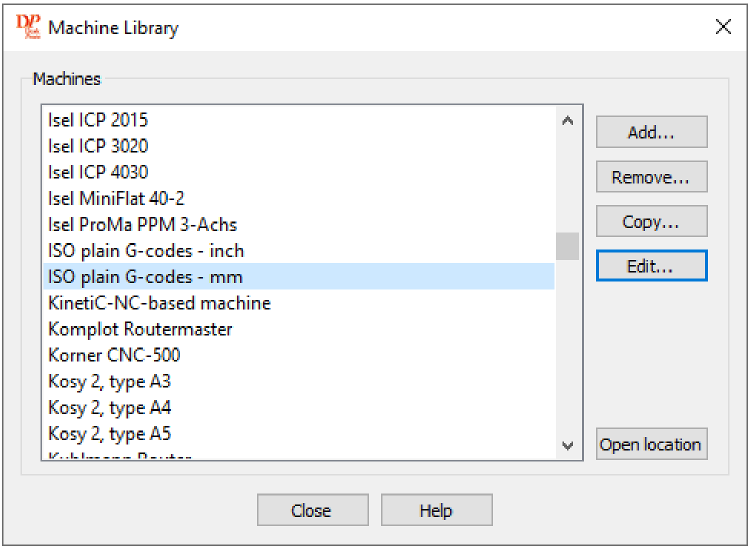
For rotation axis parts not all options are available.

3.7 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 should not match your wishes. The option used most will be the Cutter library to create new cutter definitions, as in many cases the default cutters will not match your real cutters.

3.7.1 Library of Machines



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

When you first start DeskProto you need to select one of these predefined machines as your **default machine**. This is done in DeskProto's Initial settings dialog (only visible the first time that your start DeskProto). After that you still can change this setting in the default Project parameters.
In case you have a machine that is not yet present in the list: in this Machine library you can edit an existing machine definition or add a new definition for your own machine.

You can find the Library of machines in the Options menu. When you open 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.

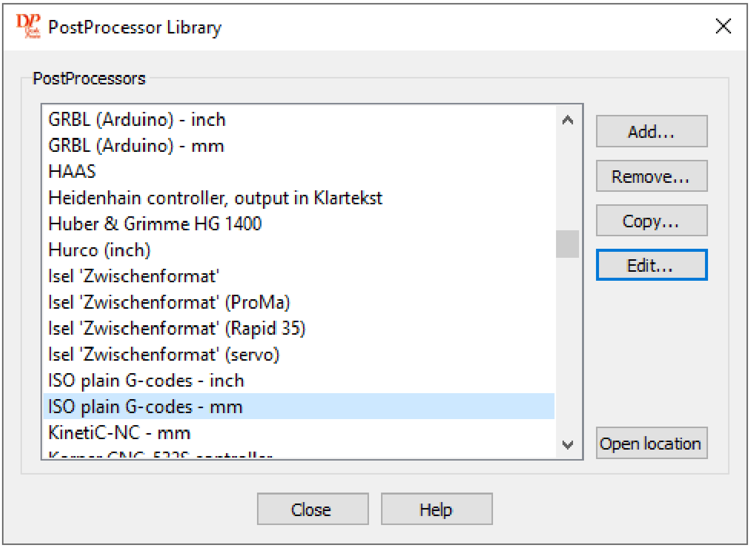
Before you start defining a new machine, be sure a postprocessor for the machine is already available. If not, configure one first in the Library of Postprocessors (see the next paragraph).

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 button "**Open location**", which will open the Drivers folder in MS Explorer, MacOS Finder or Linux File Manager.

The files are in Windows .ini format, which means that they can also be changed using a plain editor like Notepad (changing in DeskProto is safer though). A line that starts with a semicolon (;) is a comment.

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

3.7.2 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 in the format required by your NC milling machine. In computer terminology this piece of software should be called the device driver for a particular output device, however, in CNC terminology it is called a Postprocessor and we will use that name. DeskProto makes it easy to define your own postprocessor (for most CAM software this is a complicated process), 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 includes a setting for the postprocessor to be used for that machine. So a number of machines can share the same postprocessor.

You can find the Library of postprocessors in the Options menu. When you open 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 informative new name after pressing Copy.*

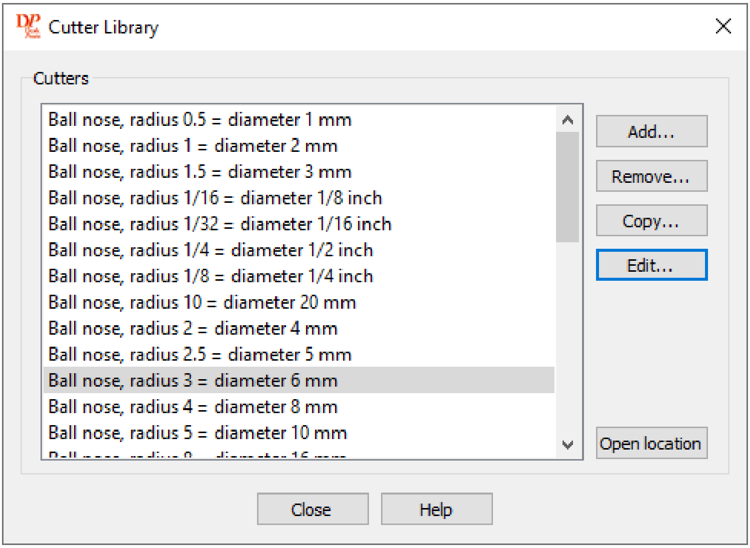
A postprocessor definition is stored as a file name .ppr in the DeskProto drivers folder (see Preferences). These files can be copied, for instance to a different PC to make the machine available on that PC as well. You can easily find each driver file via button "**Open location**", which will open the Drivers folder in MS Explorer, MacOS Finder or Linux File Manager.

The files are in Windows .ini format, which means that they can also be changed using a plain editor like Notepad (changing in DeskProto is safer though). A line that starts with a semicolon (;) is a comment.

Note 1:
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 Project Parameters dialog.

Note 2:
Postprocessor files are in a proprietary format that is defined by the software manufacturer: each manufacturer uses a different format. So these files cannot be exchanged between different CAM programs: MasterCAM for instance cannot use a postprocessor made by VCarve Pro.

3.7.3 Library of Cutters



For every NC program to be created (more accurately for every operation) you need to select a cutter. Obviously the cutter that you select for DeskProto's calculations must be available for your 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 installation. In many cases you can simply select one of the existing cutters when editing the operation-parameters. However in case you need a special cutter: in this 'Cutter Library' you can change an existing cutter or define your own.

You can find the Library of cutters in the Options menu. When you open 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 machine available on that PC as well. You can easily find each driver file via button "**Open location**", which will open the Drivers folder in MS Explorer, MacOS Finder or Linux File Manager.

The files are in Windows .ini format, which means that they can also be changed using a plain editor like Notepad (changing in DeskProto is safer though). A 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 which cutter to use: open the Operation Parameters dialog.

3.7.4 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 (so not your current project) 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 and Preserve Direction (tab Vector), Skip backfaces and Flip normals (tab Geometry) can be changed. 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 (Windows), in the Preferences section (MacOS), in the .config section (Linux). Each user has his/her own default settings stored.
As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button **Restore defaults** is present, that will restore the DeskProto defaults for all parameters.

3.7.5 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 (so not your current project). 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.

An important setting for the default part is the first operation, as that defines the **project type**. When the first operation is a vector operation the default project is a Vector project, for a geometry operation it is a Geometry project and for a bitmap operation it is a Bitmap project.

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.

Note:
The default project parameters are stored in the registry (Windows), in the Preferences section (MacOS), in the .config section (Linux). Each user has his/her own default settings stored.

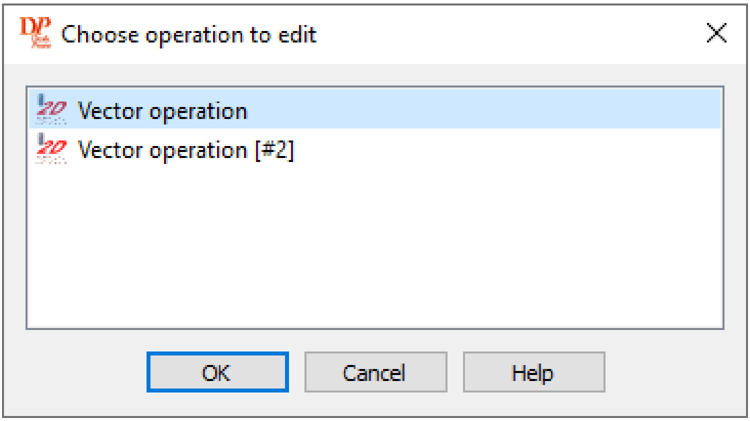
As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button **Restore defaults** is present, that will restore the DeskProto defaults for all parameters.

3.7.6 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 (so not your current project). 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 registry (Windows) / file DeskProto.plist (MacOS) / file DeskProto.conf (Linux), and will be different for every user.

In case more than one default Vector Operation have 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 that you can choose from.

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

The default project parameters are stored in the registry (Windows), in the Preferences section (MacOS), in the .config section (Linux). Each user has his/her own default settings stored.
As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button **Restore defaults** is present, that will 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.

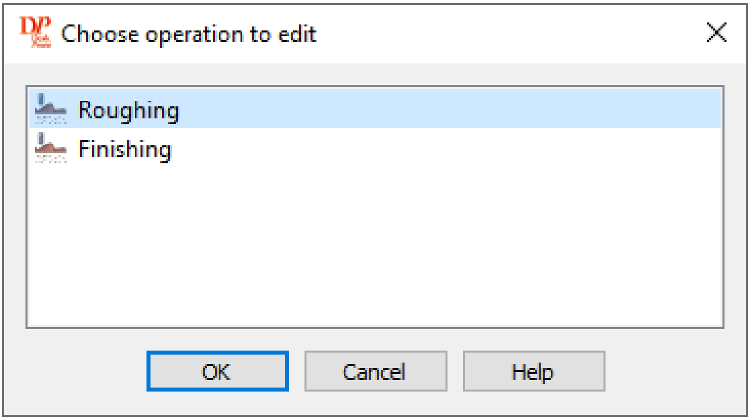
And on tab page Profiling an extra **Set...** button is present, in section Support tabs, to define the Default profiling support tab settings.

3.7.7 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 (so not your current project). 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 Geometry operation Parameters dialog, it will thus not be explained here. The difference is that this function adjusts the default geometry operation settings, that will be used for every new Geometry 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 registry (Windows) / file DeskProto.plist (MacOS) / file DeskProto.conf (Linux), 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:



In this case the dialog will of course show Geometry operations that you can choose from.

Note:
The default project parameters are stored in the registry (Windows), in the Preferences section (MacOS), in the .config section (Linux). Each user has his/her own default settings stored.
As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button **Restore defaults** is present, that will restore the DeskProto defaults for all parameters.

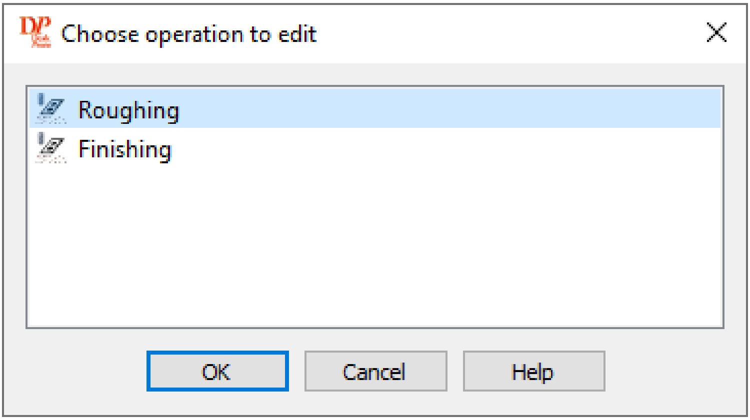
The DeskProto defaults are the built in values that were also present when you first installed DeskProto.
The default cutter might be different though: for a Geometry operation DeskProto will select the first ballnose cutter in the Cutter library with a diameter of 6 mm (metric users) or 1/4" (inch users).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

3.7.8 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 (so not your current project). 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 registry (Windows) / file DeskProto.plist (MacOS) / file DeskProto.conf (Linux), 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:

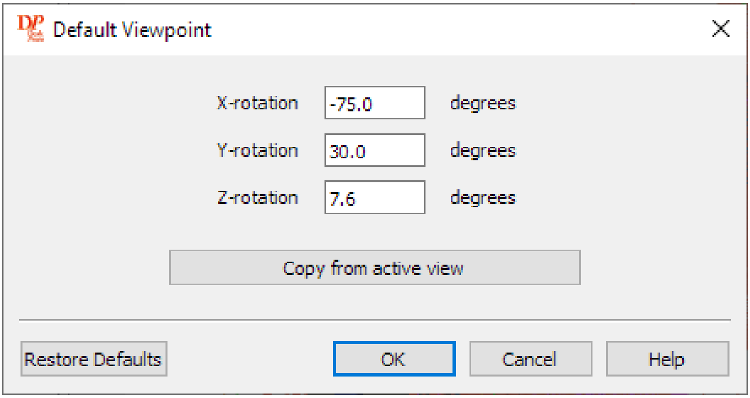


In this case the dialog will of course show Bitmap operations that you can choose from.

Note:
The default project parameters are stored in the registry (Windows), in the Preferences section (MacOS), in the .config section (Linux). Each user has his/her own default settings stored.
As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button **Restore defaults** is present, that will 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.

3.7.9 Default Viewpoint



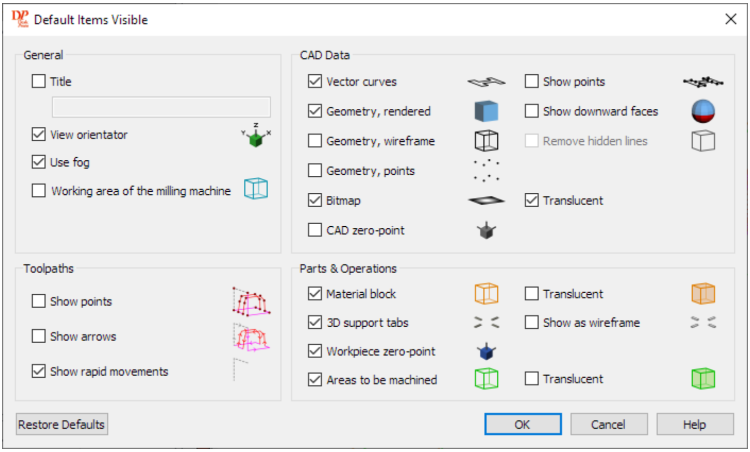
The default viewpoint settings are used for every new project and for every new part.
They are also used for the menu command and the toolbar button called Use Default View.

After choosing this option, first you will be warned that this option will only influence the defaults (so not your current view). 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 viewpoint a special dialog is used, as the dialogs to change the Viewpoint contains parameters that are not suitable as default parameters: here only the rotation can be set. Zooming of the default view will always be 100% and 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 button **Restore defaults**.
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 viewpoint settings are stored in the registry (Windows), in the Preferences section (MacOS), in the .config section (Linux). Each user has his/her own default settings stored.

3.7.10 Default Items visible



The default items visible settings are used for every new project and for every new part.

After choosing this option, first you will be warned that this option will only influence the defaults, so not your current View window. 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 items visible a special dialog is used, as the dialogs to change the Items visible contains parameters that are not suitable as default parameters.
Here it is possible to set what will be shown initially. You can find an explanation for any of these 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.

Note:
The default items visible settings are stored in the registry (Windows), in the Preferences section (MacOS), in the .config section (Linux). Each user has his/her own default settings stored.

3.7.11 Preferences

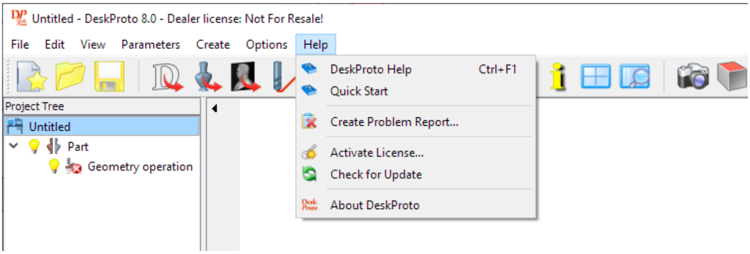
This is open the Preferences dialog, in which you can edit DeskProto's preferences. The dialog consists of six tab pages.

In Windows and in Linux this command can be found in the Options menu, in MacOS in the DeskProto menu.

In **Windows** these preferences are stored in the Registry
in **MacOS** in file *~/Library/Preferences/com.delft-spline-systems.DeskProto.plist*
in **Linux** in file *~/.config/Delft Spline Systems/DeskProto.conf*

Note that in MacOS a file called *~/Library/Preferences/com.delft-spline-systems.DeskProto80.plist* is present as well. That is a different file, automatically created by MacOS to remember the settings for the Open file dialog and the Save file dialog.

3.8 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).

3.8.1 DeskProto Help

This command will open the DeskProto Help system, shown in a separate dialog.

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 (it may be needed to first open the line "DeskProto Reference" to see these five lines):

- 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 on the DeskProto website), which contains almost exactly the same information as the Help file.

In addition to the Contents tab page three more tabs are available in the Help dialog:
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.
Other methods to access Help are:

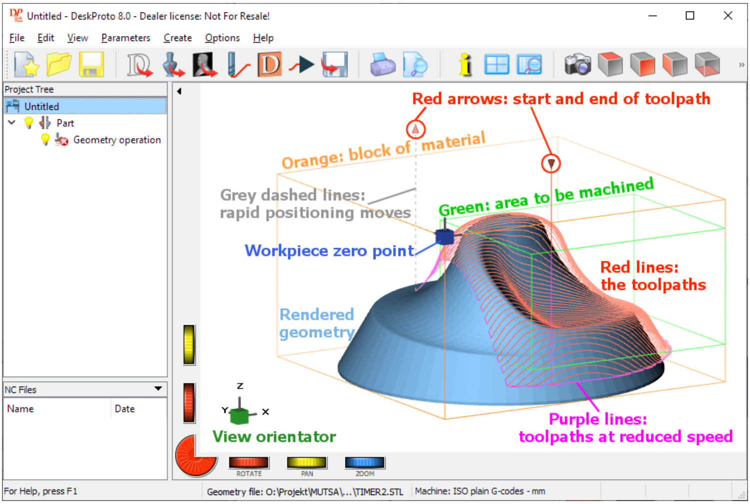
- The Help button on the Toolbar: 
- Pressing the F1 Function key on the keyboard (for MacOS this is Fn+F1).
- **Every dialog has a Help button, providing Help information about the use of that particular dialog.**

3.8.2 Quick Start

This help page is shown when you select command "Quick start" in the DeskProto Help menu.

Normally we advise to read about how to start using DeskProto in the **Tutorial book** (either printed or as PDF download). This manual step by step introduces 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. As an alternative you can find many **instruction videos**, both on the DeskProto website and on YouTube.

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: you will need this information to be able to understand what is happening. After that, novice users are advised to first use the Wizards: these will guide them through all steps needed to generate an NC program file.
In all cases do not forget to use the **Help file**, which will explain every DeskProto screen in detail.

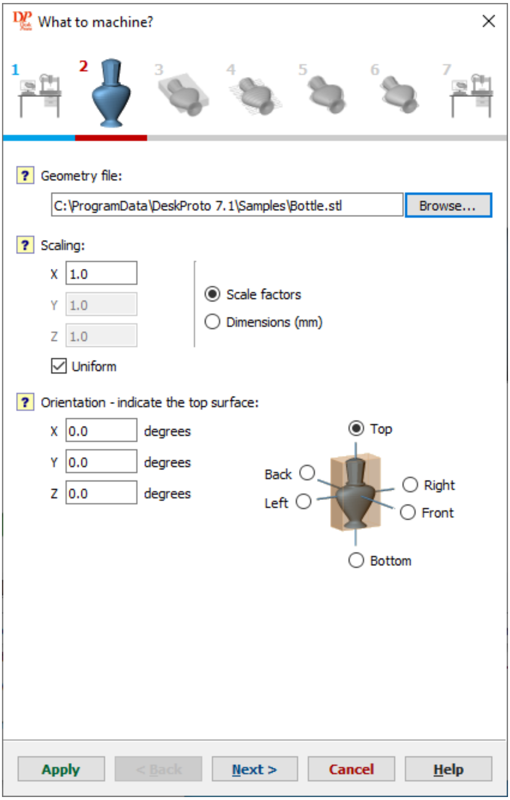


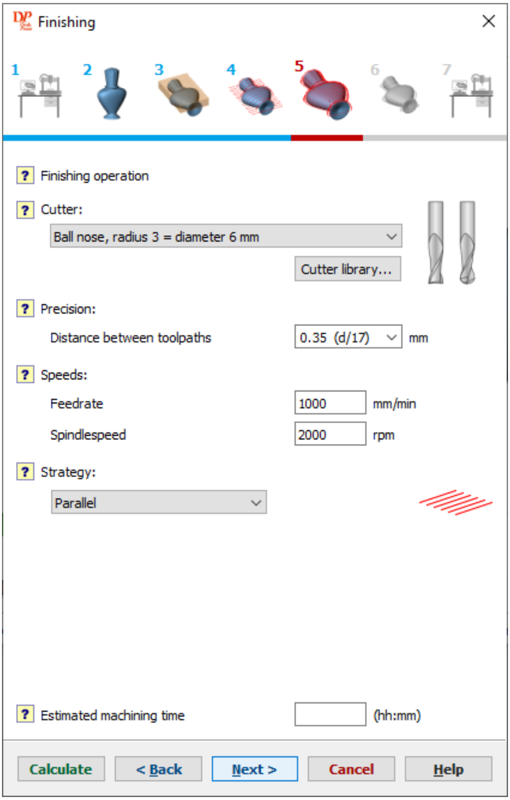
The DeskProto screen contains standard items like the title bar (top line), menu bar, toolbar (the row of buttons below the menu) and status bar (bottom line). The center area is divided into three tiles: the large View window on the right, and the windows Project Tree and NC files on the left. Follow these links for more information on any item.

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

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

It is important to know that DeskProto offers two different user-interfaces: the **wizard-based** interface and the **dialog-based** interface.


What to machine?
1
2
3
4
5
6
7
Geometry file:
C:\ProgramData\DeskProto 7.1\Samples\Bottle.stl
Browse...
Scaling:
X 1.0
Y 1.0
Z 1.0
Scale factors
Dimensions (mm)
Uniform
Orientation - indicate the top surface:
X 0.0 degrees
Y 0.0 degrees
Z 0.0 degrees
Top
Back
Left
Right
Front
Bottom
Apply
< Back
Next >
Cancel
Help

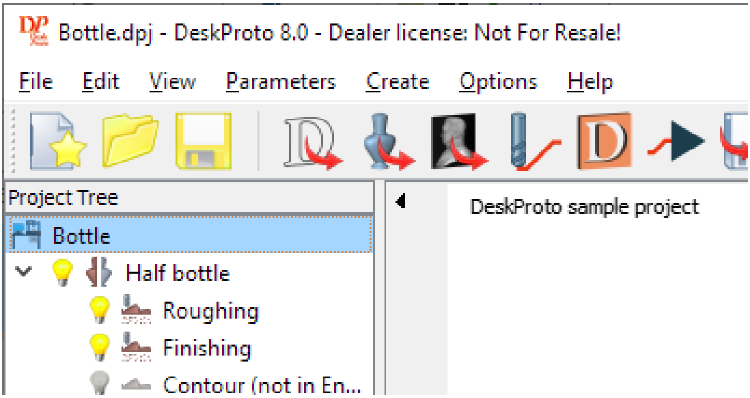


[1] Wizard-based interface.
New users are advised to use the DeskProto Wizards, as these will guide them through all the steps needed to generate an NC toolpath file using their own CAD-data. The illustration above shows two typical wizard pages. A wizard will set the same parameters as are available in the dialog-based user-interface, only now they are presented in a sequential series of screens, and only the most important parameters are shown. 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 the part: **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 an operation: **How the part 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 (by clicking on the lamp) 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 three most important menus are:

* 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 buttons in the toolbar.

* 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 (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 resulting part, to check your toolpaths for correctness.

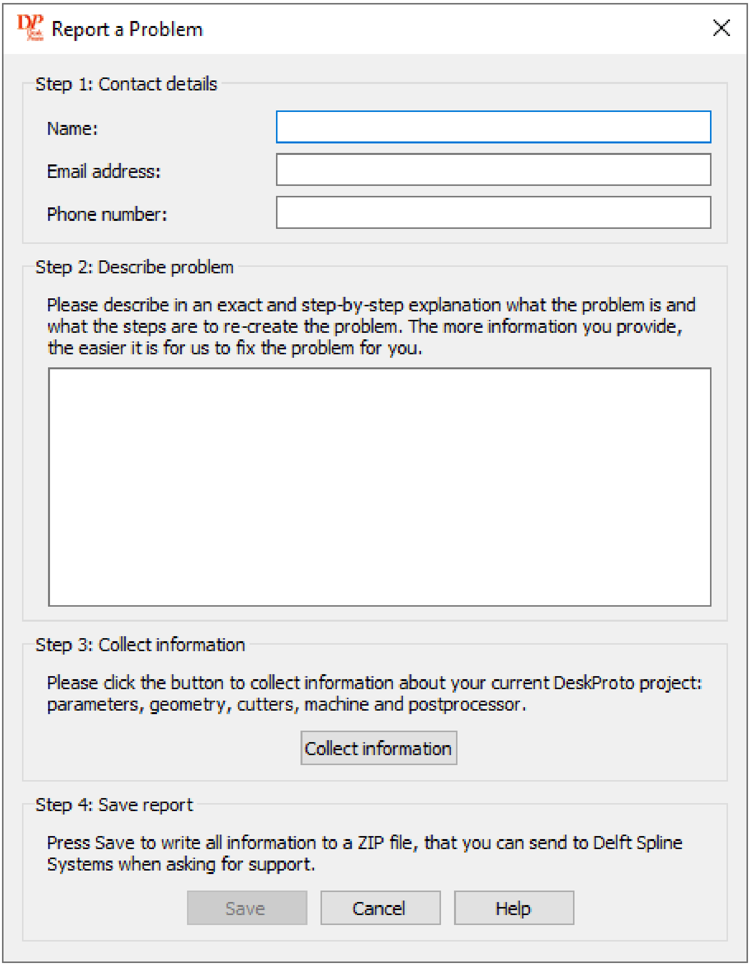
Optionally you can show an **Animation** that shows how the cutter moves along the toolpath.

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

We hope that you will enjoy using this software !

3.8.3 Create Problem Report

When asking for support, DeskProto users often send us the DeskProto project file (DPJ file), planning to make us see what they are doing. Unfortunately this is not sufficient: the DPJ file only contains the parameters that you have set. And in order to reproduce the problem the CAD file(s) are needed as well. 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 (do this when your problem project is open):

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

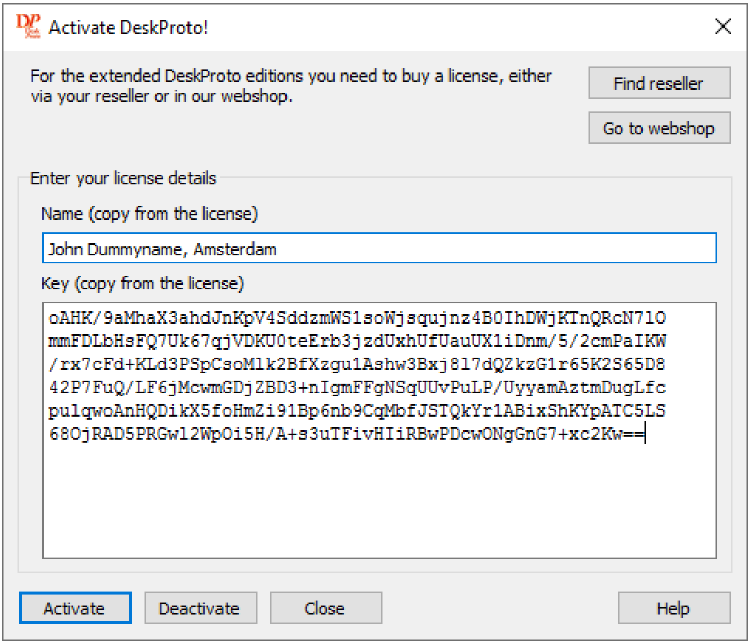
In Step 2 you are asked to **describe the problem**, in plain text. The more clearly you describe the problem, the more chance that we can indeed reproduce it. And that is absolutely needed in order to start looking for a fix.
Note that you cannot actually use DeskProto while this Problem Report dialog 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 first save this file (in case needed).

Step 4 it to finally **save 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 when saving.

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.

3.8.4 Activate license



In Windows and in Linux this command can be found in the Help menu, in MacOS in the DeskProto menu.

The **Free** Edition of DeskProto can and 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**.

In case you are not sure which edition you need: one of the options in the Free edition is to evaluate (trial) these higher Editions - when trialling a watermark (the "Trial cross") will be visible in all machined parts. On the DeskProto website you can also find a comparison table for these four editions.

In all cases you need to first download the DeskProto installation file from our website www.deskproto.com and instal DeskProto on your computer. This installation file (a Setup file for Windows, a DMG file for MacOS or an AppImage file for Linux) 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, as log as you like.

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 your 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. Even the smallest 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** your license: you will again need Name and Key when (for instance) you buy a new computer.

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

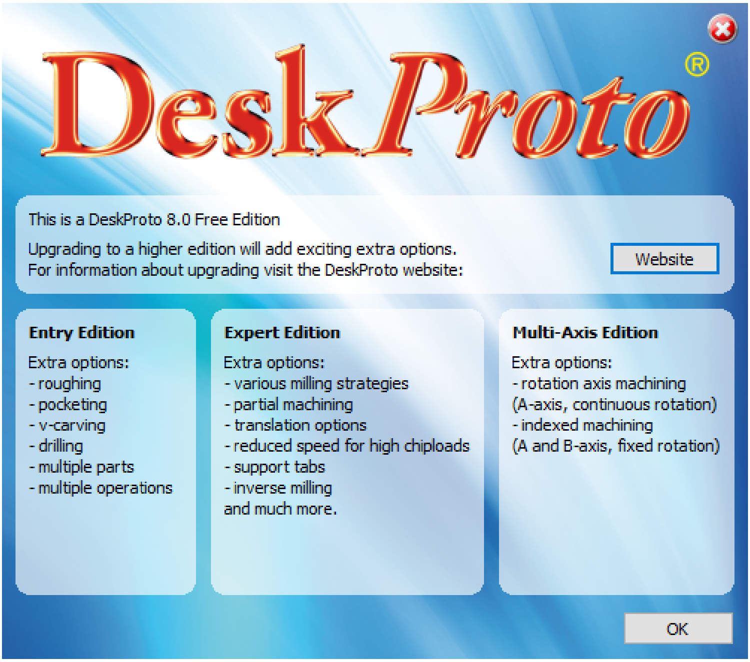
After Activating it is also possible to **Deactivate**, allowing you to again convert the program from paid license to free license. Needed for instance when you sell your computer.
You can Activate and Deactivate as many times as you like: information about the previous licenses will simply be deleted.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

Both for Activating and for Deactivating **administrative privileges** are required (Windows).

In **Windows** the license activation affects all users of that PC, in **MacOS** and in **Linux** the activation information can be accessed only by the current user: every next user will also need to again activate the license on that same computer.

3.8.5 Upgrade information

In Windows and Linux this command can be found in the Help menu, in MacOS in the DeskProto menu.
The command is present only when you use either the Free edition, the Entry edition or the Expert edition.



Displays a dialog screen that lists the advantages of the higher DeskProto Editions

DeskProto is available in four different Editions:

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

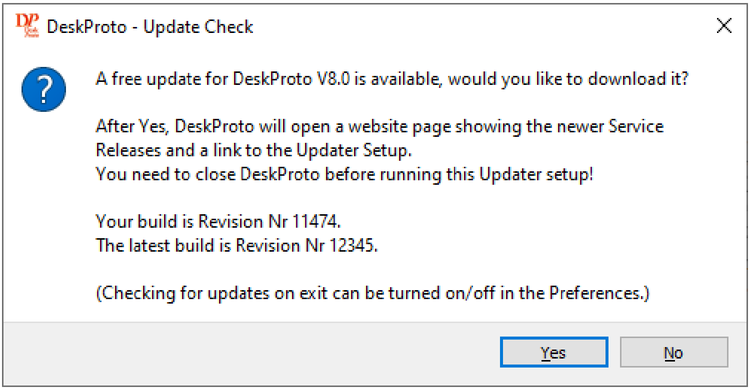
The difference between the editions is the number of parameters that you can set (both Part parameters and Operation parameters), and (for the Free edition) the number of Parts and Operations.
The difference between the Expert Edition and the Multi-Axis edition is that more axes are available in the latter: the A-axis and B-axes rotation axes.
On the DeskProto website you can find a complete comparison table.

Of course for registered users of DeskProto upgrades are available at a special upgrade price.

3.8.6 Check for Update

This command will make your DeskProto contact the website www.deskproto.com (an internet connection is required) to see if a newer Service Release (**bugfix build**) and/or a newer version is available for download. In Windows and Linux this dialog can be found in the Help menu, in MacOS in the DeskProto menu.

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.php



The command Check for Updates will automatically compare the Revision number of your DeskProto with the Revision number of the latest release, it will tell you when a newer release is available for download, and it will ask if you want 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.
For MacOS and for Linux no special Updater Setup is possible: for these operating systems you will need to download a new installation file.

In addition to these **Bugfix updates** DeskProto also checks for 'minor upgrades' and for 'major upgrades', and will show information if these are present.

For DeskProto 8.0 a **Minor upgrade** would be the upgrade to V8.1. When that has been released it will also be a free upgrade for users of V8.0 as it will recognize the same license.
For DeskProto 8.0 a **Major upgrade** would be the upgrade to V9. When that has been released it will by a paid upgrade for users of V8.0, available at a special upgrade price.

When you are using the latest release DeskProto will tell you so.
And when no internet connection is available DeskProto will tell you that it "Could not perform the Update check".

DeskProto by default will **automatically perform this update check** each time you exit DeskProto. For this automatic check these last two messages (about using the latest release and about being unable to check) are suppressed.
You can switch off this automatic check in the Preferences.

3.8.7 About DeskProto

In Windows and Linux the About command can be found in the Help menu, in MacOS in the DeskProto menu.

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

Version number, like 8.0

Edition - so either Free, Entry, Expert or Multi-Axis, and if the program is 32 bits or 64 bits.

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

Serial number - Each new (paid) 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

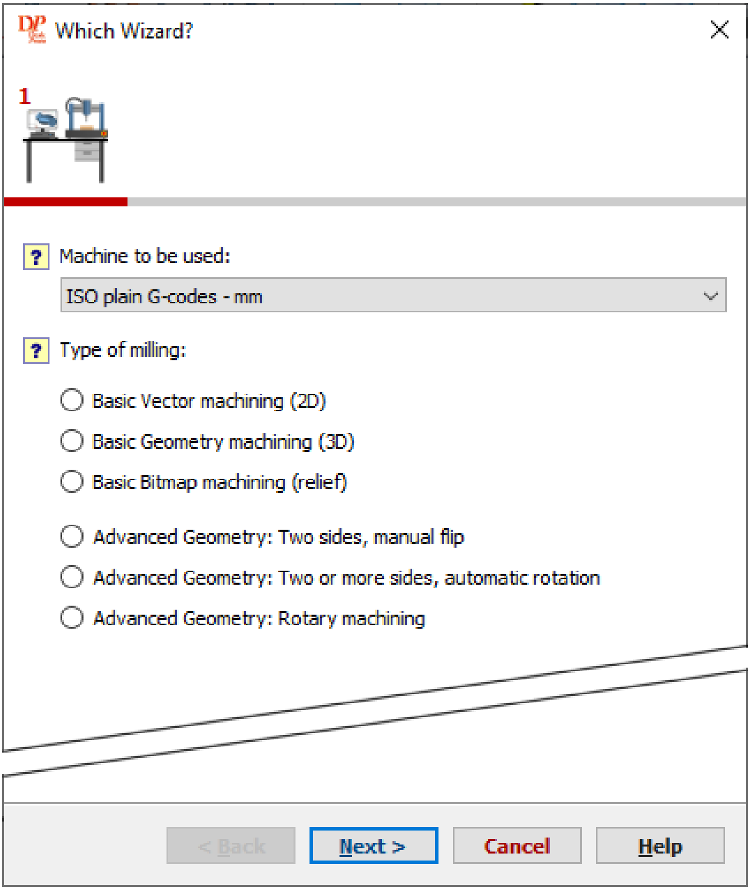
Contact information - DeskProto is made by Delft Spline Systems in the Netherlands

Short program description.

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

IV Dialogs

4.1 Wizards



The Wizard-based user interface is an important feature of DeskProto. It makes it possible for users without much CAM know-how to easily create the toolpaths that they need for their projects. Each wizard is a series of dialog screens (pages) that need to be filled in: the wizard so to speak *'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.

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 its own Help information page.

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

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

Calculate will calculate the toolpaths 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, SVG). It is meant for novice DeskProto users, and explains this procedure step-by-step. You can choose to use either Profiling, Pocketing or V-Carving toolpaths.

Basic Geometry Machining
This wizard creates 3D toolpaths for just one Geometry file (STL, 3MF, 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.

Two or more sides, automatic rotation, also called the **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 XYZ 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 all around: rotation axis machining. In contrast to the previous wizard, now the material rotates during machining (continuous rotation).

A rotation axis is an extra piece of equipment that lets your material rotate on the machine (just like a roast rotating on a spit above a barbecue).

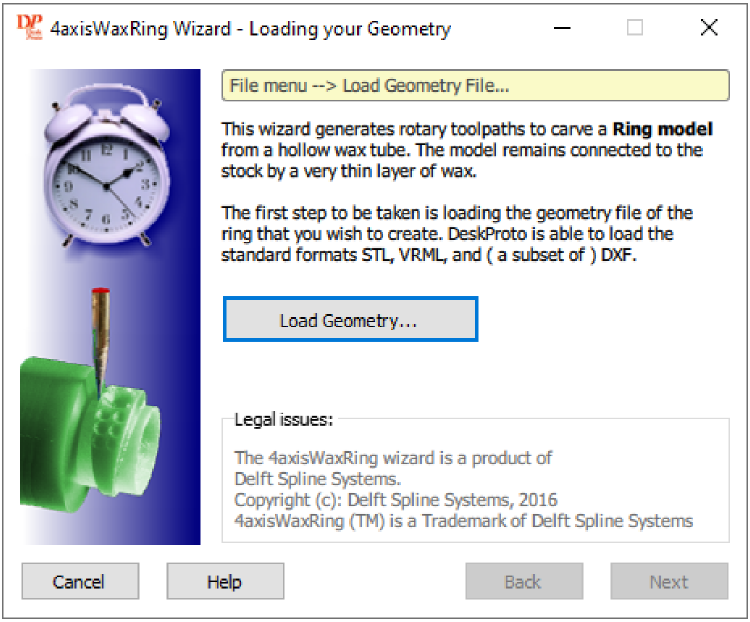
Note that all functionality offered by the wizards is also available in the dialog-based user interface: the wizards are only meant to make things easier for you, they do not add new options. After finishing any wizard you can still use the dialogs to fine-tune the settings that the wizard made.

You can find the Wizard in the File menu (File >> Start wizard) or in the Start Screen.

Shortcut:

Keys: Ctrl+W (Windows, Linux) ⌘+W (MacOS).

4.1.1 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: for 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: Cancel, Help, Back and Next. The meaning of each button will be clear.

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.

Custom wizards are not supported in MacOS and in Linux.

4.2 Parameters

4.2.1 Project

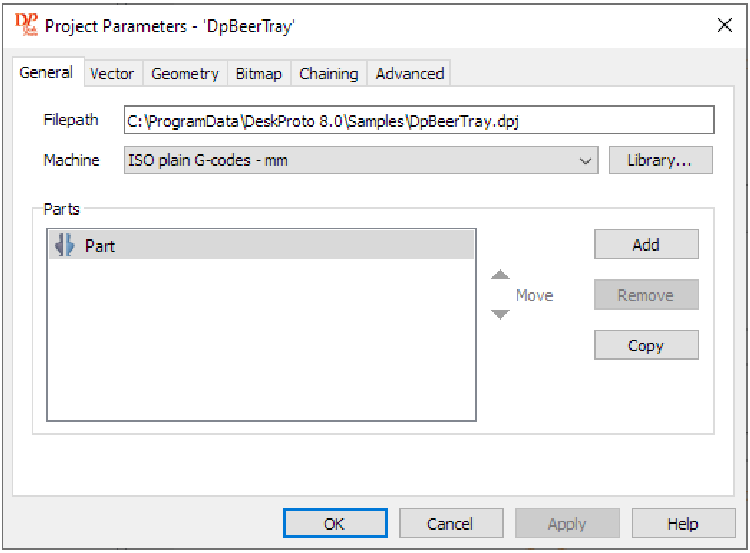
4.2.1.1 Project Parameters

The Project parameters dialog offers all settings that are available on project level. Most important are the CNC milling **machine** and the CAD **data files** to be used for this project. Each of the six 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

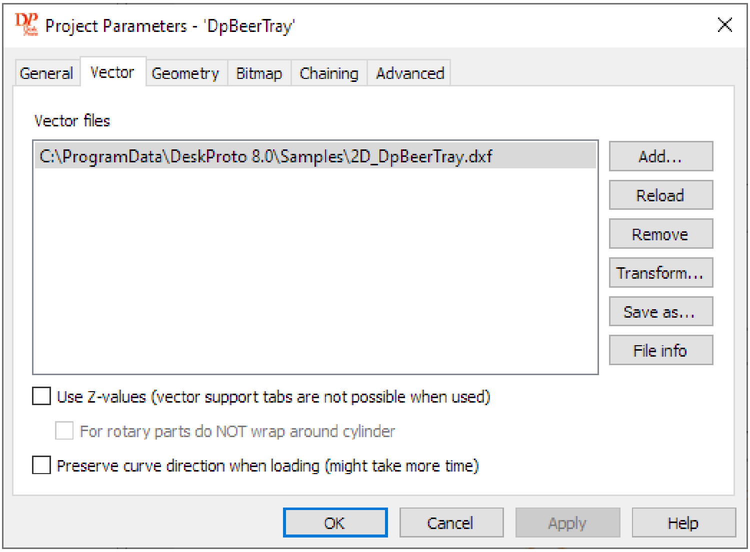
This filed shows the path and name of the currently loaded Project file. This information cannot be changed in this dialog. To start a new project or to save the project using a different name, use the File menu.

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 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 (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 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 (gray background color): the buttons on the right will influence that file (except of course the Add button).
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 it has again been exported from CAD) you can **Reload** it.
Very powerful is the **Transform** option, which will open the Vector Transformation dialog: 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, number of points, dimensions.

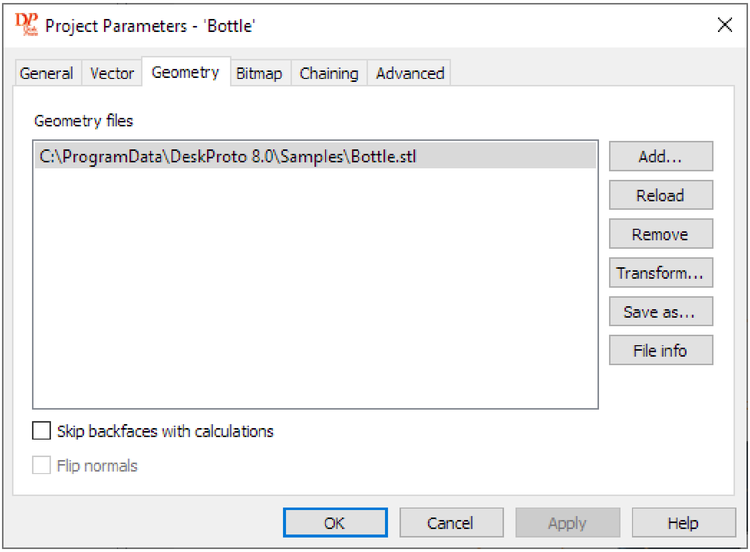
Use Z-values
Vector machining in most cases will be done using two-dimensional CAD data: a drawing that does not contain any Z-values (only X and Y, to be shown at the top of the block). Nevertheless DeskProto will also accept vector files that contain Z-coordinates. When opening such file DeskProto 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 (or point) as it has been 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 (in CAD coordinates)for all points on the vector curves. This will influence the result when both vector files and geometry files are loaded: the CAD zero point of the geometry file then will be used for the curves, where normally the curves will be drawn at the top of the block.

For rotary parts do NOT wrap around cylinder
Using vector data in a rotary part in most cases is about a text or a drawing that needs to be engraved on the part. DeskProto achieves this by wrapping the 2D vector drawing around the cylinder shaped block (for a rectangular block around the smallest possible cylinder that encloses the complete block). The cutter then always is oriented perpendicular to the 2D drawing, just as for 3-axis machining.
For vector files that use Z-values this is not always the intended behavior though: the 3D vector curve may exactly follow the geometry: in order to machine a groove in the surface or to cut through sheet material. In such cases you can check this option. The curves then will not be wrapped, and the toolpath will simply follow the curves in 3D.

Preserve curve direction when loading
When loading a vector data 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 applies a small tolerance (0.001 mm), 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 (gray background color): the buttons on the right will influence that file (except of course the Add button).

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 has again been exported from CAD) you can **Reload** it.

Very powerful is the **Transform** option, which will open the Geometry Transformation dialog: 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, number of points, dimensions.

Skip Backfaces

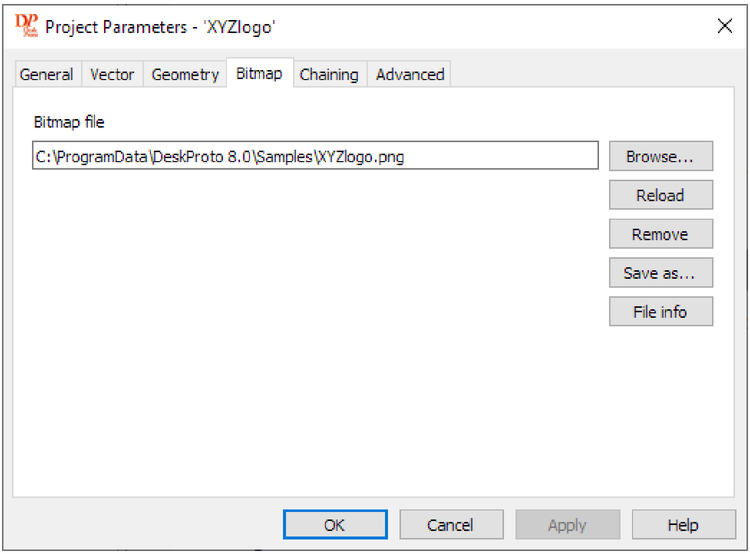
In an STL file for each facet (triangle) a normal vector is stored, indicating which side of the facet is on the outside of the geometry and which side is on the inside. The inside triangles are called backfaces. For valid STL file these are all completely invisible, as an STL geometry should define a closed solid.

The option Skip Backfaces saves calculation time, as it makes DeskProto simply skip all backfaces. This means that half of the triangles will be skipped, which results in shorter calculation time. The resulting toolpath will be the same (for a valid geometry). If this option is selected all backfaces are colored bright red in the drawing that DeskProto displays - for a correct STL file no red will be visible. As nowadays almost any STL file is valid this option is checked by default.

You can clearly see the red facets in sample file Teapot.stl, meaning that this is not a valid STL file. The defined surface does not fully enclose a some space, so this teapot is not a valid solid. Which is not surprising for a dataset modeled back in 1975 (by Martin Newell at the University of Utah): the term solid modeling then did not yet exist. For corrupt geometries like this you may need to uncheck this option to skip the backfaces.

Flip Normals
As just explained, in DeskProto for each facet (triangle) a normal vector is stored, indicating which side of the facet is on the outside of the geometry. 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. This is an old option, nowadays it is needed only very rarely, as currently all new STL files are valid.

Bitmap parameters



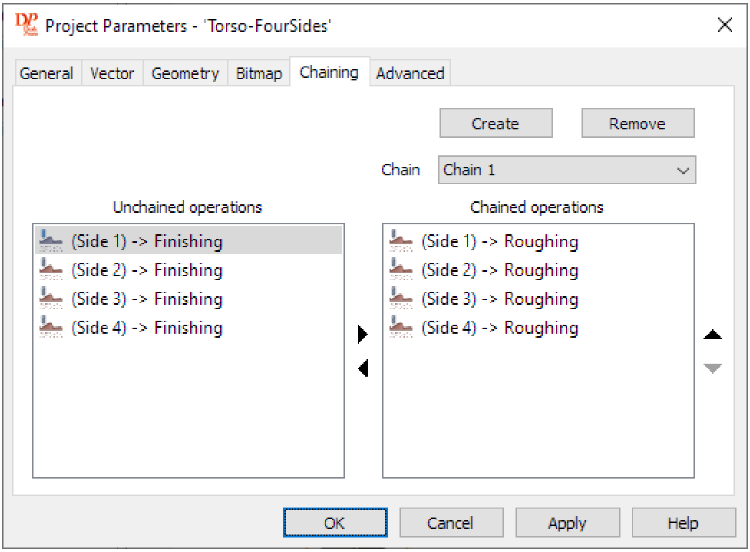
Bitmap file

This tab page shows the Bitmap file (if any) being used in this project (for bitmap data only one file can be loaded in a DeskProto project).
You can change this file using the **Browse** button, on the right, and you can 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.

The number of options is lower than for Vector files and for Geometry files, as you can use only one bitmap file in your project.

For bitmap files no extra options are available.

Chaining parameters



Chaining is an option that allows you to combine Operations from **various Parts** into one large NC Program file. Normally only Operations in the same Part will be combined into one NC Program file. 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 possible only in case more than one part is present in the current project.

Also note that Chaining is an option for advanced users: only apply this feature this when you know how to use it.

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

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

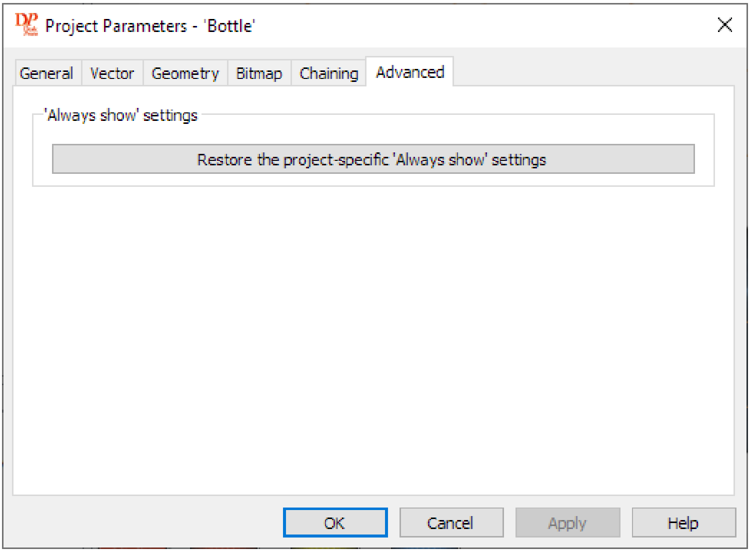
The combo box **Chain** shows the currently selected chain (if any), and makes it possible to select other chains. Chain names are automatically generated: Chain 1, Chain 2, etc.

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 (needed for indexed machining).

Advanced parameters



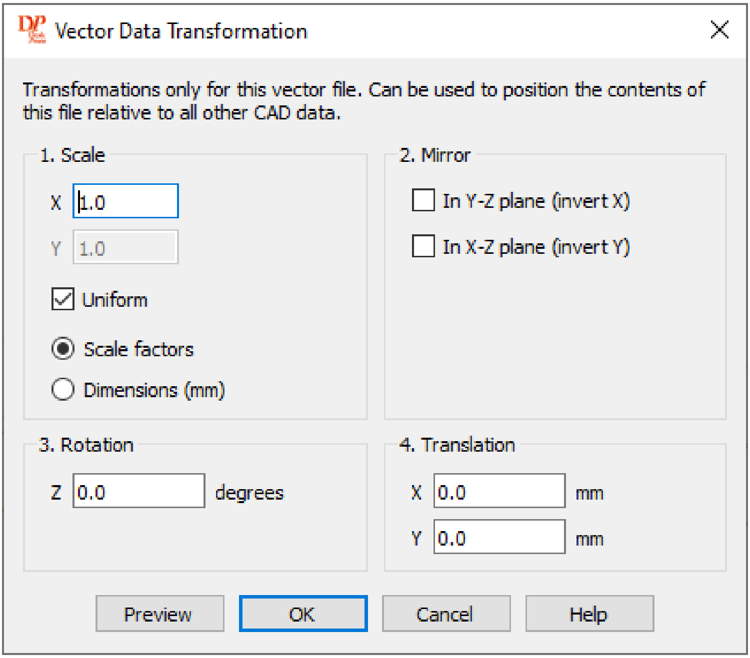
Many warning messages in DeskProto have a checkbox in the left bottom corner: *'Always show this message'* or *'Always show this message for this project'*. As default this checkbox is checked: the warning will be given when needed. When you uncheck the box the warning will no longer be displayed.

For *'Always show this message'* the warning will remain suppressed as long as you use DeskProto. Some only for the current session, others even when you again start DeskProto some time later.
For *'Always show this message for this project'* the warning will only be suppressed for this project, also when later reopening the project (setting stored in the DPJ file). This is called a project-specific Always show setting.

In case you want to undo suppressing the warning you cannot simply check this box again, as the warning no longer is given. You can use this button **Restore the project-specific 'Always show' settings** to achieve this, for all project-specific warnings.

The other warnings (the general Always show settings) can be restored via a button on the Advanced tab of the Preferences, in sections 'Settings'.

4.2.1.2 Vector Data Transformation



DeskProto 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 here, as you can give each file its 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.

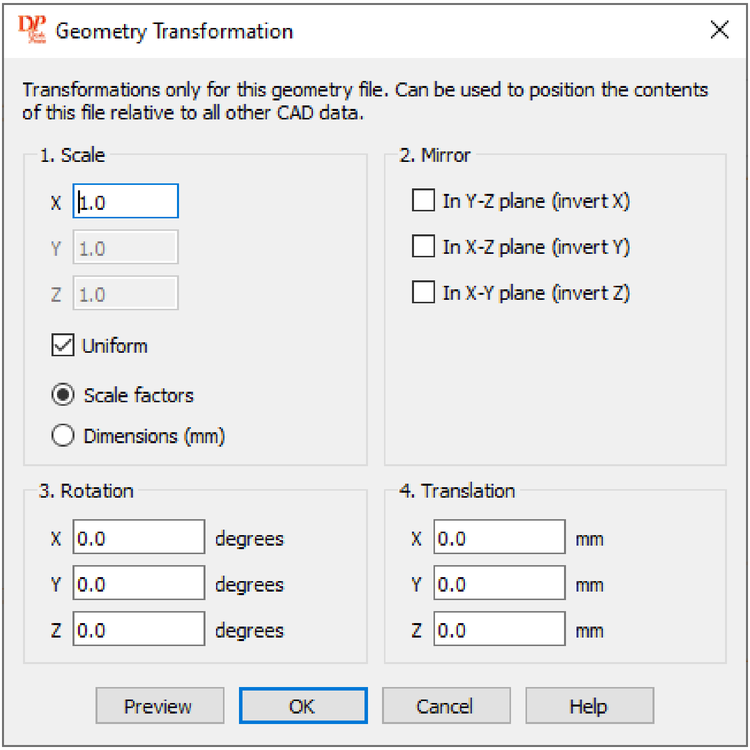
This dialog can be opened on the Vector page of the Project parameters, using button Transform....
It will also pop up when loading an extra Vector file, and enables you to position the new vector data relative to CAD data that is already present. It offers basically the same transformations as also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

Parameters (see there for more help), however these will be applied only to this one vector file.

A 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 what you need, you can enter new transformation values and again press Preview for a new drawing. When all is OK you can acknowledge with OK.

You need to realize that in the drawing that you see the Part Transformations will be applied as well, **after** the transformations that you enter here. This means that (for instance) when in the Part Parameters a rotation of 90 degr. round Z is applied, a translation along Y that you enter here will result in a translation along X on your screen..

4.2.1.3 Geometry Transformation



DeskProto allows loading more than one Geometry file (and more than one Vector data 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.

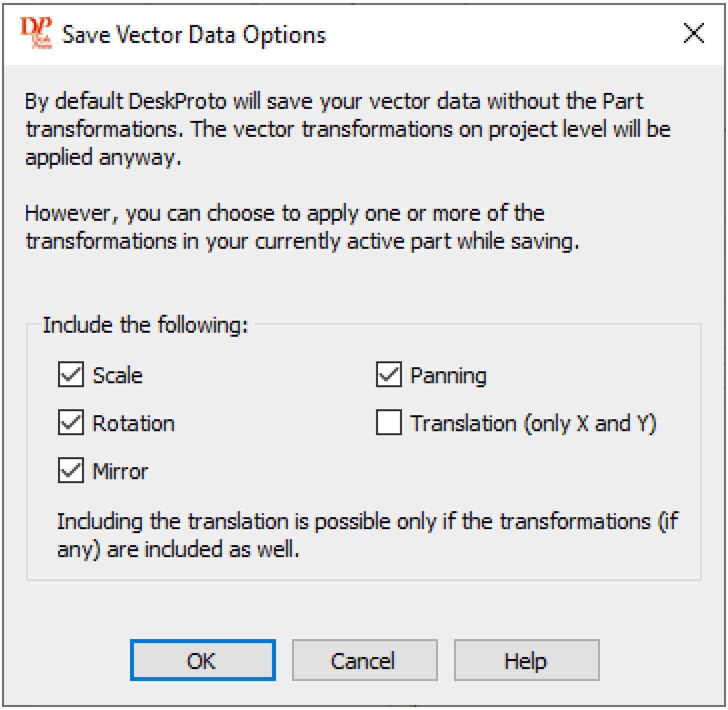
This may result in unwanted collisions and/or overlaps. Such unwanted situation can be corrected here, as you can give each file its own set of transformations. For instance a translation to position two (or more) identical geometries next to one another.
In the **Geometry Transformation** dialog these 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 only to this one geometry file.

A 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 what you need, you can enter new transformation values and again press Preview for a new drawing. When all is OK you can acknowledge with OK.

You need to realize that in the drawing that you see the Part Transformations will be applied as well, **after** the transformations that you enter here. This means that (for instance) when in the Part Parameters a rotation of 90 degr. round X is applied, a translation along Y that you enter here will result in a translation along Z on your screen..

4.2.1.4 Save Vector Data Options

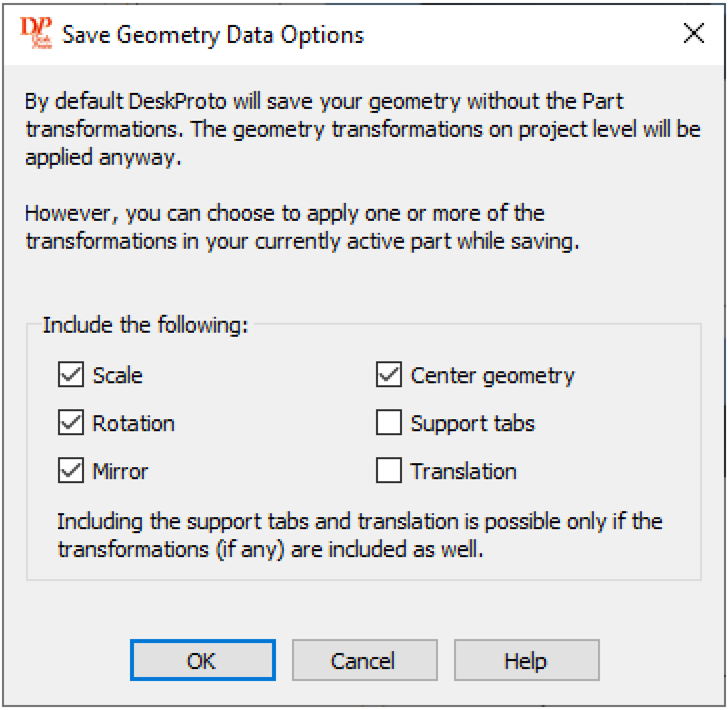


The **Save Vector Options** dialog enables you to apply (in the new file) the Transformations that you have set in the Part Parameters. It will pop up after pressing command Save Vector Data As... in the File menu, and after pressing the Save as... button on tab Vector of the Project parameters.

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.
All Vector Data Transformations that have been set for each file on Project level will also be applied when exporting a Vector data file, these cannot be switched off in this dialog.

In case multiple vector data files have been loaded: when calling this command *from the file menu* the total of all vector curves will be exported.

4.2.1.5 Save Geometry Data Options



The **Save Geometry Options** dialog enables you to apply (in the new file) the Transformations that you have set in the Part Parameters. It will pop up after pressing command Save Geometry Data As... in the File menu, and after pressing the Save as... button on tab Geometry of the Project parameters.

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.
All Geometry Transformations that have been set for each file on Project level will also be applied when exporting a Geometry file, these cannot be switched off in this dialog.

In case multiple geometry files have been loaded: when calling this command *from the file menu* the total of all geometries will be exported.

4.2.2 Part

4.2.2.1 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, because 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 tab 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.

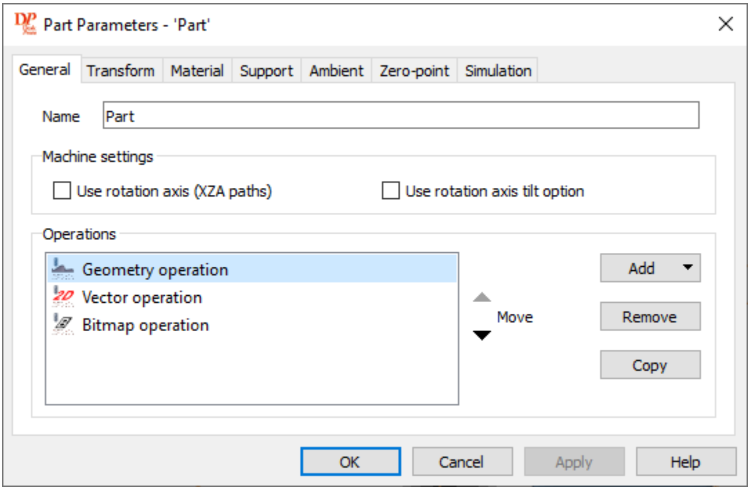
The Part Parameters dialog has an extra button: **Apply**. Using this button you can apply any new setting immediately, without having to first close the dialog. Any changes will be immediately reflected in the drawing on screen and in the Part information dialog.

This dialog can be reached via the Parameters menu.
Shortcuts:
You can double-click on a part line in the project tree (one of the second level items).
Or right-click on a part line 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: that will make it easy to interpret the Project Tree when you have defined more than one part. This name is for your convenience only: it is not used in the resulting NC program file (unless you add it in a comment line using a postprocessor placeholder).

Machine settings

These are visible and/or enabled only in the Multi-Axis edition of DeskProto. The first checkbox only for machines with a Rotation axis (4th axis), the second checkbox only for machines with a 5th axis or with a manual Tilt option. The manual tilt option is very rare: only a few machines support this.

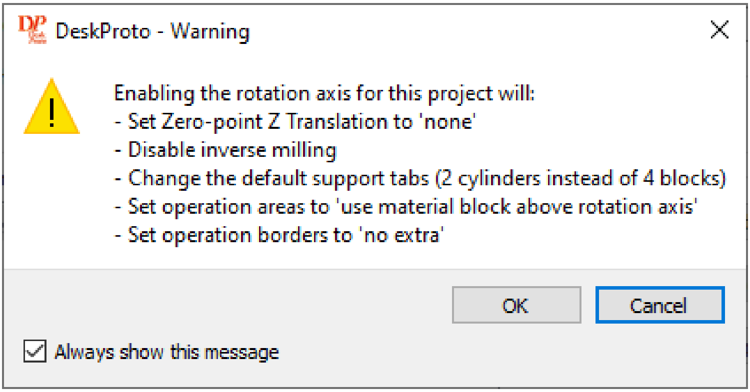
Use rotation-axis

A rotation axis is a device on the machine that rotates the part during machining. Like a roast 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 cutter.

By default DeskProto generates standard XYZ toolpaths, and thus ignores the rotation axis.

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 operation area no longer is a rectangular block: it becomes a cylinder instead.
The part will rotate when machining: continuous rotation. This in contrast to the indexed machining, which is possible using DeskProto too, in the wizard "Two or more sides, automatic rotation".

For geometry-data DeskProto will rotate the 3D part when calculating the toolpaths, bitmap data will be wrapped around a cylinder. For vector data both options are possible, in the project parameters you can set which option to use.



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 as that does not work for rotary machining.
- change the default setting for the Support Tabs.
- set the Operation areas to "Above rotation axis", in order not to let the tool sink below the rotation axis.
- set the Borders to "No extra" for all operations in that part (as the part must not be cut loose of the rotation axis)

You are free to change these settings later in case needed. For instance sometimes the cutter needs to go below the rotation axis.
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.

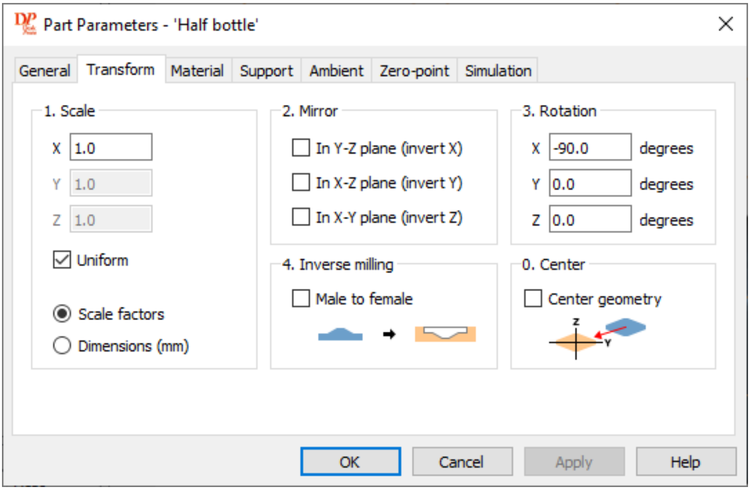
Use 5th axis as rotation axis tilt option
Use rotation axis tilt option
This option is available only for some machines.
In case a 5-axis machine (with a CNC controlled 5th axis) has been selected the first text is displayed: Use 5th axis as rotation axis tilt option.
In case a machine has been selected that supports the manual tilt option the second text is displayed: Use rotation axis tilt option.
This option is available only in the DeskProto Multi-Axis edition. If this box is not checked DeskProto will ignore the tilt option and generate standard toolpaths.

The manual tilt 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 tilting option is a B-axis rotation. However, 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.
On trunnion-style 5-axis machines (with a CNC controlled 5th axis) the large rotation axis can be used to tilt the small rotation axis that is mounted on top, and then rotate the small rotation axis while machining.

Operations
The number of operations within the part can be changed by adding, removing or copying operations, using the buttons **Add**, **Copy** and **Remove**. Three different types of Operation can be added: Vector, Geometry and Bitmap, so 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 of options 4 and 0 on this tab page will differ per edition of DeskProto (Free, Entry, Expert, Multi-Axis).
The availability of 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. For instance scaling is done before rotation, so when you rotate you may need to scale a different axis than visible on the screen.

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. 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.
The image above shows the scale factors with one decimal, DeskProto will accept up to 5 decimals (five digits after the decimal point, like 1.12345): accurate scaling may be needed for instance to exactly compensate for product shrink.

Note that 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.

2. Mirror
The mirror option is the second transformation that is applied. It is only useful to mirror in one direction, as mirroring in two directions is identical to one rotation, and 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 symmetrical 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 that is applied. Note the difference between this rotation (which changes the part) and the view rotation (which only changes the viewpoint / the camera position). Both rotations use an identical process, 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!) such block or area can remain correct only for rotations over (multiples of) 90 degrees. For other angles the boundaries would be changed, so these are not permitted. For Freeform areas only rotations round X and Y over (multiples of) 180 degrees and round Z over (multiples of) 90 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.
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 (one of the first four translation options). The latter condition is always true in the Free edition and the Entry edition, as there the translation cannot be set.

The button **Align to...** leads to the Align dialog: a handy help to set a few useful locations.

0. 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. As 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.* As a result the block of material may become far larger than the geometry. The complete geometry then may even 'orbit' the rotation axis at a large distance.

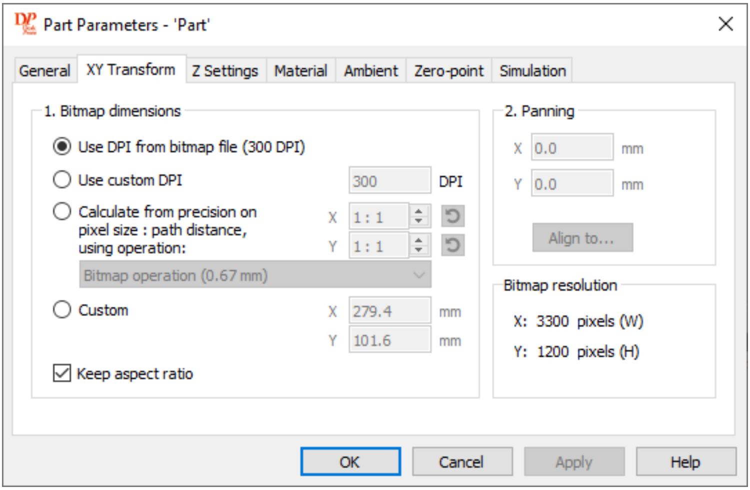
When you check this option DeskProto will translate the geometry to a position where the origin is in the exact center of the geometry.
DeskProto will still rotate around the X-axis, however this axis now will go through this centerpoint. You can immediately see the difference as this influences the size of the Material block: when checked it will exactly fits the geometry. The Center Geometry option is performed before the other transformation, hence it has been given sequence number 0.

When you need any other position of the rotation axis you can apply a custom translation in the Geometry Transformation dialog, as that translation will be applied before any part transformations. Or you can of course 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 after calculating 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 it sets 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 an exact size for 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 **Moiré pattern** (see paragraph Bitmap Data), like a small ridge every few mm. Which can be fixed by choosing this option (approximate the preferred 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, directly in mm or in inches.

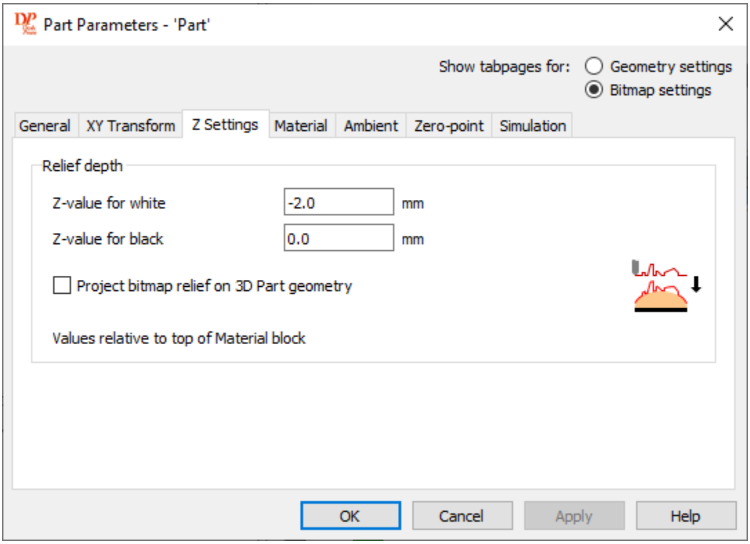
The option **Keep aspect ratio** takes care that the ratio between X size and Y size is not changed: image and relief will have the same XY ratio. So when you change the X size with this option checked, you will see that the Y size automatically changes too.

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) in CAD coordinates, 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 (one of the first four translation options). The latter condition is always true in the Free edition and the Entry edition.

Z Settings parameters

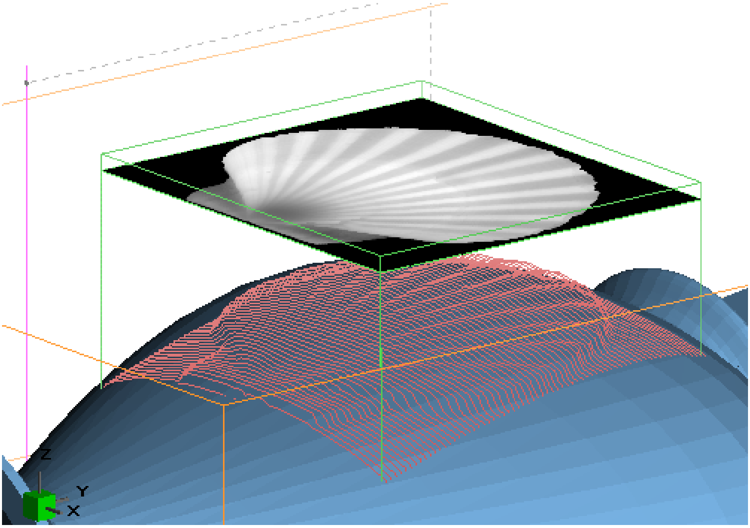
(Bitmap Settings)



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

Relief depth
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).

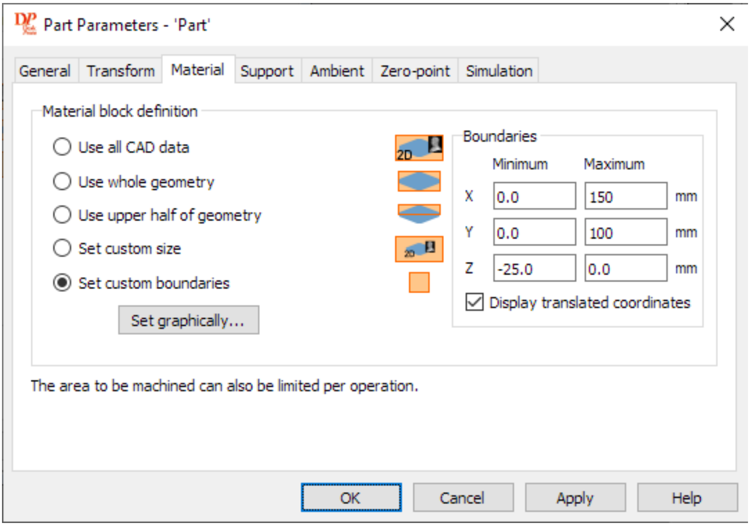


Here a shell relief is created from a bitmap image, and projected on the perfume bottle that you have seen more often as example part. This is of course a great design tool to create embossed products. The picture shows a positive relief: on top of the geometry.
Important for such positive relief is 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 block needs to be high enough as well: the bounding box of the STL file in this case will be too low.

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.
This option is activated only when a geometry has been loaded in the project, as of course otherwise projection is not possible.

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).

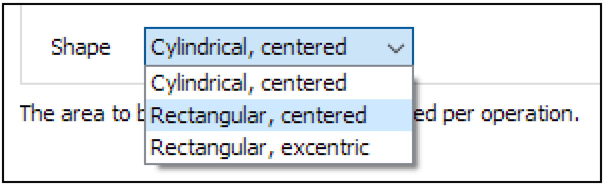
The illustration above shows this tab page for 3-axis machining a 3D geometry. In other situations other options may be present:
- the options that mention Geometry are available only when a geometry file has been loaded.
- the option "Set custom size" comes with only three edit fields, for the X, Y and Z dimensions.
- when rotation axis machining is selected the option "Use upper half of geometry" disappears, two extra edit fields are present for the Min and Max R-values (Radius), and an extra option is present to set the Shape of the block (see below).

Four or five 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 create one model. This option is not present for rotation axis machining.
- **Set custom size** is the option to define a different block size. You can enter the block size by typing the appropriate dimensions for X, Y and for Z. The CAD data will be located in the center of the block.
- **Set custom boundaries** gives you more control when setting a different block size, as you can now enter the block dimensions by typing the minimum and maximum boundaries. You can always reset the values by again selecting 'Use all CAD data". As said the Min and Max R-values are available for rotation axis machining. It is also possible to change the boundaries using your mouse, which can be done after pressing button Set Graphically.

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 the 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.

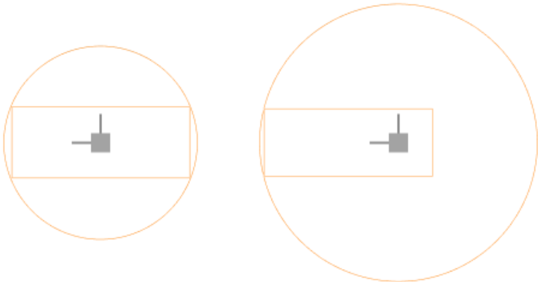


When rotation axis machining has been selected you can set the **Shape** of the block. Three shapes are possible:

- **Cylindrical, centered**: the block is a cylinder, with the rotation axis as center line. Only the X-boundaries and the R-boundaries can be set. A minimum R lower than 0.0 is not permitted. When you enter a minimum above 0.0 the result will be a tube-shaped block (hollow).
- **Rectangular, centered**: a rectangular block, with the rotation axis as center line. Only the X boundaries and the max Y and Z boundaries can be set. Min Y will be minus Max Y, same for Z
- **Rectangular, eccentric**: a rectangular block, Now all six X, Y and Z boundaries can be set, so minimum and maximum can be set at different

distance to the rotation axis. The rotation axis needs to be inside the block though.

For three-axis machining and for indexed machining only rectangular blocks are possible.



When changing the shape of a custom block from rectangular to cylindrical DeskProto needs to calculate new boundaries (using R instead of Y and Z). This conversion is done using the CAD zero point as center for the cylinder. So when that CAD zero point is not in the center of the geometry the cylinder will become large, as shown in the drawing above.

Using custom material block size in most cases will mean that 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.

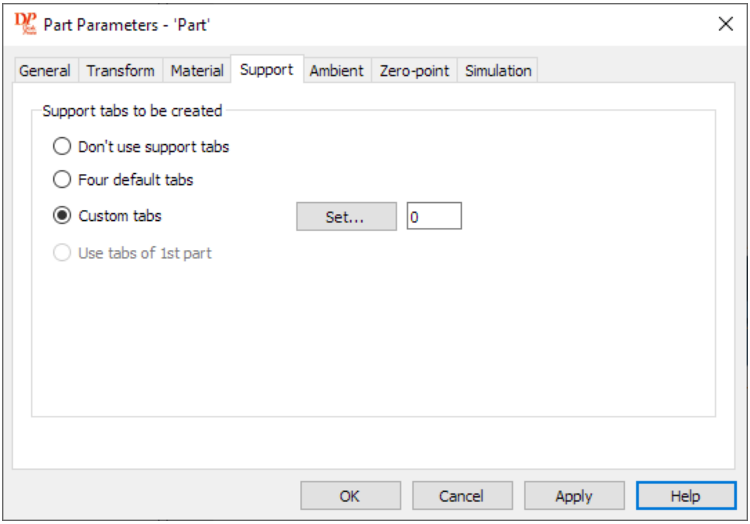
The same trick can be used when your part is too high for your machine: build the model from separately machined slabs of material. For instance produce a model of 140 mm high, milling three separate slabs of standard tooling board blocks of 50 mm thick, using three parts with these material blocks: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



When your material block is much larger than the part you can add **support tabs** to keep the part connected to the rest of the block. These are small blocks of material (various shapes are available), acting as connection bridges to hold the part on its place. You then can clamp the block on its corners and machine several parts, each on a different position, without the need to clamp each separate part.
Tabs are also needed for two-sided machining (see the Two Sided Milling Wizard), and for fixturing your model on a rotation axis device (to keep the part connected to the rotary table).

This page is about Geometry support tabs, created 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:\ProgramData\DeskProto 8.0\Supports\
The contents of this STL file are scaled and rotated as needed, and then are added to the part as a special type of geometry, drawn in a different color. The difference with all other geometry is that the area below the tab is

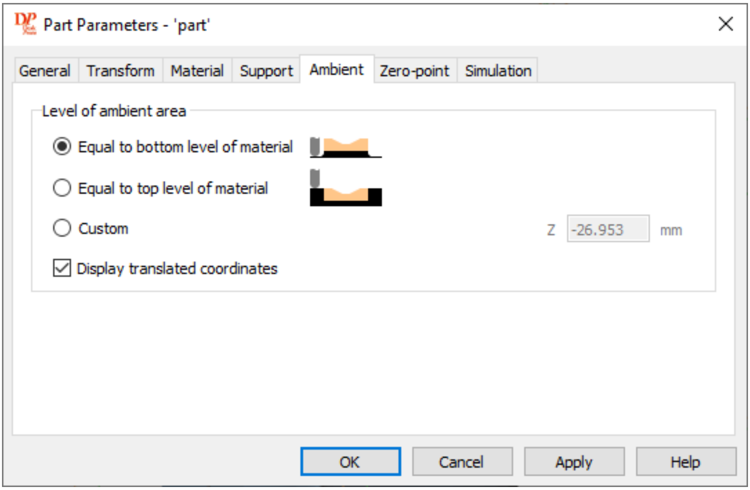
treated as ambient area when Skipping ambient, so DeskProto will not create a toolpath around the tab but move over the tab instead. This also effects the Contour only strategy: the cutter will rise and travel over the tab instead of round it.

Four options are available:
- **Don't use** support tabs, which is the default and will be OK for most parts.
- Use **Default tabs**: DeskProto will locate these tabs at the extreme points of the geometry (min and max X, min and max Y, applied after the rotations). For normal XYZ machining four rectangular tabs are generated (parallel to X and Y), for rotation axis machining two cylinder shape tabs (parallel to and centered around the rotation axis). Tab width and thickness depend on the size of the geometry, tab length is (at least) 1.5 times the cutter diameter (10% of this length 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 create any number of custom tabs, using button **Set...**. This will open the Set support tab dialog, which offers options to graphically set the position, size, shape and orientation.
- **Use tabs of first part** is very convenient when your part is machined from several sides: DeskProto will then create one "part" for each side, and for each parts the same support tabs are needed. Define these support tabs only once (for the first part), and use that same definition in all other parts. Advantage is that any changes need to be made only once. This option is very handy for two-sided machining. For the first part obviously this option is disabled.

In order to keep support tab connected to the surrounding material it is needed to select one of the Ambient skipping options (Operation parameters, tab Advanced), as otherwise the end of the tab still may be cut loose.
An alternative solution is to make the end of the tab stick outside the Area to be machined. When the end of the support tab is exactly on the edge of this area you need to also **switch off the borders** in each of the operations used for this part (Operation parameters, Tab page Borders).

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.

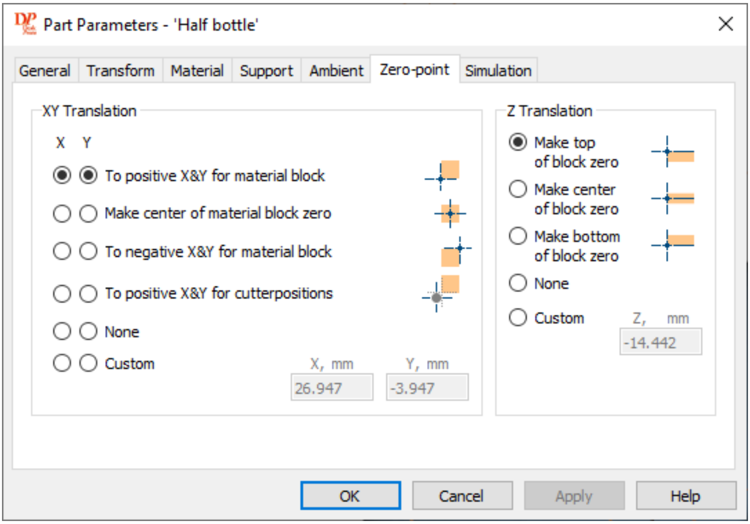
- The default choice is **Equal to bottom level of material**, so at the minimum Z-level of the block. This will do for most cases, as then all material around the part will be removed. **Note**: when using this option, for **ballnose cutters** the actual Z-level will be R (radius of the cutter) *lower 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 toolpaths: converting from CAD coordinates to machine coordinates. Although you can see translation as a form of transformation, it has not been placed on the Transform tab page of these Part parameters. In DeskProto the translation is applied after all actual calculations, just before saving the toolpaths to an NC-program. So in the Part parameters it is shown as last tab that influences the toolpath data.
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 though, as the cutter needs to move outside the block to machine the outer surfaces.

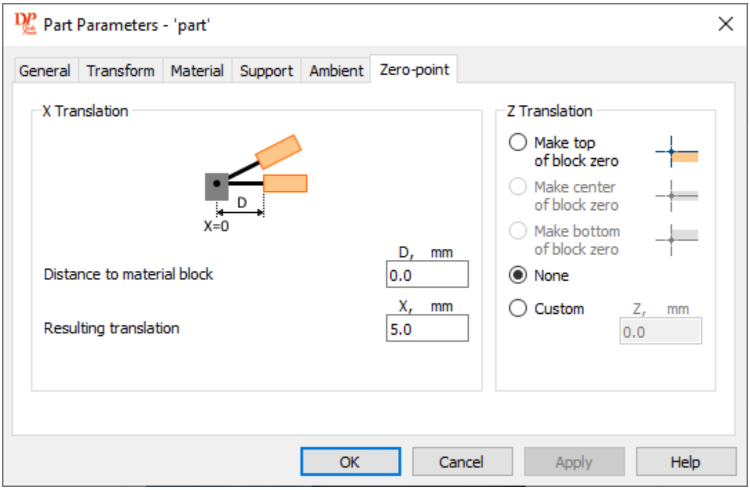
It is possible to use different settings for X, for Y and for Z. For each axis five or six predefined translation options are available. For each option DeskProto will calculate the actual translation value and show it in the edit box after option Custom.

XY:
- Translate **To positive X&Y for material block** has just been explained: X=0.0 resp 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.
- **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.
- Translate **To negative X&Y for material block** will set the X=0.0 resp Y=0.0 on the other side of the block. In some cases that is more convenient than the first option.
- 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). Extra attention is needed for this setting, as when using this option toolpaths for various cutter diameters do not share the same zero point.
- **None** means that no translation is applied. The edit box will show 0.0, the zero-position of the CAD data 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 boxes. The value that you enter is the translation between CAD coordinates and workpiece coordinates. The edit boxes as said also show the actual translation for any of the other options, however the value then cannot be edited.

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 explanation on the Ambient page.

- **None** means that no translation is applied. The edit box will show 0.0, the zero-position of the CAD data 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 one of the options **Use 5th axis as rotation axis tilt option** or **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 this rotation point 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.

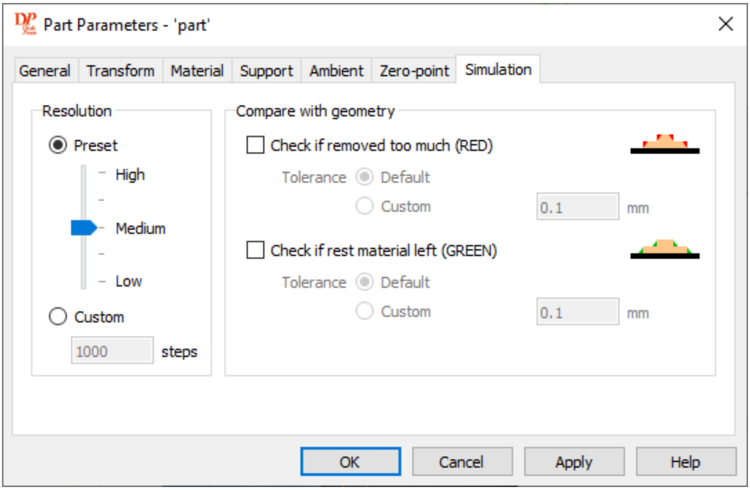
The **Resulting Translation** will be different from the Distance to material block, as the zero point of the STL file (CAD zero) in most cases is not located on the left edge of the block.

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.

The illustration above shows the situation when on tab General the option **Use rotation axis** has been checked as well: in that case not all **Z Translation** options are possible.

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 **Resolution** sets the accuracy of the simulation to be calculated.
The 3-axis simulation (XYZ) applies a Z-grid, similar to 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. We call this the **grid-based simulation**.

The rotary simulation uses a completely different algorithm, based on voxels (very small 3D cubes). Here the level of detail sets the number of voxels that are used. The more voxels the more accurate the result (smaller cubes), and the longer the calculation time. We call this the **voxel-based simulation**.

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

	Grid-cells	**Voxels**
Low	200	200
	500	400
Medium	1000	700
	2000	1200
High	5000	2000

This number of grid-cells or voxels is used for the longest side of the block to be simulated. The number of cells along the short side of the block is calculated proportionally. The number are lower for voxels than for grid-cells, as voxel calculations need far more computer resources.
In most cases the default setting (Medium) 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. In addition, calculating a high level of detail will also need much memory.

You can enter a **Custom** resolution in case one of these five presets does not match your needs. For this custom resolution as well two values are present: one for Grid based simulations and one for Voxel based. DeskProto will automatically convert this value when you check or uncheck the option "Use rotation axis" on the General tab page.

Compare with Geometry offers the possibility to check the difference between the resulting part and the original STL file geometry. 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.
This option is of course only active for a Geometry project. It is also not possible for a rotary simulation.

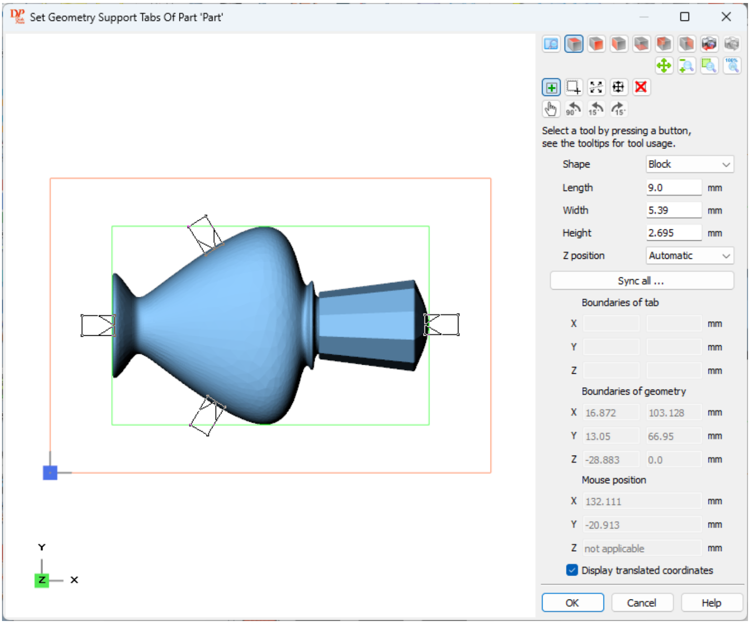
Check if removed too much (RED) will show a red color in case DeskProto has removed too much material. Note that the red surfaces will only be visible when you have turned off the rendered geometry in the Items visible dialog, otherwise the geometry will hide the red.

Check if rest material left (GREEN) will show a green color 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

- small holes 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.

4.2.2.2 Set Geometry Support Tabs



The Set support tab dialog offers options to add, edit and/or delete geometry support tabs. You can reach this dialog by using the **Set** button on the Support tab of the Part parameter dialog or on the Material and Supports page of one of the Geometry wizards.
In order to make a support tab effective:
- either the end of the tab needs to be outside the Area to be machined (like in the image above: the green line shows the area),
- or you need to select one of the Ambient skipping options (Operation parameters, tab Advanced).
Otherwise the end of the tab may be cut loose.

The dialog shows a **top view** drawing of your part, with all support tabs (if any) drawn with lines. The bounding box of the tab is drawn, and the arrow indicates the attachment point of the tab (important for tab shapes that are not symmetrical).

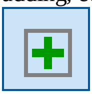
Most of the functions can be accessed using the buttons in the top-right corner of the dialog, We will explain the functionality of these buttons below, row by row. Note that when you hover the cursor over a button a **tooltip** will pop up showing relevant information.

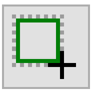
Buttons in row 1
Eight of these are standard DeskProto buttons to set the viewpoint, of which the use is known (as explained on page Toolbar) and needs no explanation here.

 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.

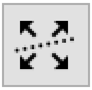
Buttons in row 2
These four buttons (on the right side) are standard DeskProto buttons as well. These are connected to the buttons of rows 3 and 4, as of all buttons on these three rows (except Zoom all) only one is the active button, indicating the current mouse function. That active button is shown with a blue background color. The meaning of the four buttons in this row will be clear: Mouse panning, Mouse zooming, Mouse zoom window and Zoom all. Mouse rotation is not present: this dialog only uses the six main views.

Buttons in row 3
The buttons in this row are for the main function of this dialog: graphically adding, editing and deleting support tabs.

 **Add new tab**. This button will be the active button when you open the dialog, as the first thing you will want to do is create one or more tabs. Point the cursor to the location where you want to add a support tab (on the outside edge of the geometry) and then click the left mouse button. The tab will be oriented perpendicular to the closest outside edge of the geometry (in top view), or if that is not possible it will be oriented along the line from the click-point to the closes geometry point. Shape, dimensions and Z position are conform the settings shown below.

 **Draw new tab**. This button also adds a tab, now by drawing its bounding box rectangle. So its orientation always is parallel to X or to Y, and

its size is set by the rectangle. Only the Z-position comes from the settings below.



Resize or rotate tab. This button offers a lot of functionality to edit a tab. When it is active the mouse offers these functions (*the cursor shows which of these functions is active*):

When clicking on a corner point of the tab you can **resize** the tab.

Keeping Shift pressed means to keep the aspect ratio of the tab

Ctrl+Shift pressed means that either the length or the width of the tab will be changed, depending on your mouse movement (on a Mac use the Command key instead of Ctrl).

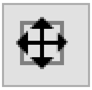
When clicking near a corner point of the tab you can **rotate** the tab.

Keeping Shift pressed means to rotate in steps of 15 degrees

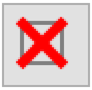
Ctrl+Shift pressed means steps of 5 degrees, and

Ctrl pressed means steps a 1 degree.

When clicking on a line of the tab you can **move** that line.



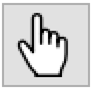
Move tab. This button allows to pick up the complete support tab and move it to a different location.



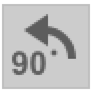
Delete tab. With this button you can delete a tab. Note that when clicking the tab is immediately deleted, without first asking for a confirmation, and without an undo option.

Buttons in row 4

The four buttons in this row offer extra functionality.



Select tab. The tabs in the dialog are drawn with black lines. One of the tabs can be selected: the selected tab is drawn in blue lines. When a tab is selected the settings shown below the buttons concerns this selected tab. You then can change the selected tab by editing these settings.



Rotate tab 90 CCW. The three rotate buttons are active only when one of the tabs is selected. This button will rotate the selected tab 90 degrees, Counter Clock Wise. Rotations are possible only round the Z-axis, see the Note at the end of this page for an exception.

Rotate tab 15 CCW. This button will rotate the selected tab 15 degrees, Counter Clock Wise.

 **Rotate tab 15 CW**. This button will rotate the selected tab 15 degrees, Clock Wise.

The five lines below the buttons make it possible to set the Shape, the Size and the Z-position of a support tab. These settings are applied when the button Add tab is pressed, or - when a tab is selected - they can be used to edit that tab.

The **Shape** of the tab can be set using a combo-box (button with a drop-down menu). As default five shapes are available: Block, Cone, Cylinder, Pyramid and Wedge. Default shape is a Block. The Cone, the Pyramid and the Wedge are asymmetric: one end is much smaller than the other end. The tab will be connected to the geometry with its smallest side, making it easy to finish the part after removing the support tabs. It is possible to add other shapes, see the Note below.

The **Length**, the **Width** and the **Height** are the dimensions of the tab - at least for a block-shaped support tab. For the other shapes these are the dimensions of the tab's bounding box. When changing the dimensions of the selected block, the center point of the block will remain the same. The default dimensions are:
Length: 1.5 times the diameter of the largest cutter that is used in the operations of the part. 10% of this length sticks inside the geometry, the resulting 1.35 times the cutter diameter offers sufficient room for the cutter to move round the part.
Width: 10% of the smallest size (X or Y) of the part
Height: half the width (however always smaller than the height of the block).
The support tab by definition is horizontal (so both ends have the same Z-value)

The **Z-position** of the block cannot be seen in the top view that this dialog shows as a default: change it to front view or side view to see the-position. Here as well a combo-box button, offering the options Top, Centre, Bottom and Automatic.

Button **Sync all...** offers the option to set Shape, Length, Width, Height and/or Z-position for all support tabs. The pre-set values in this dialog are the current values for a new tab, or in case a tab is selected the values of the selected tab.

Boundaries of tab

Here you can see the minimum and/or maximum values for X, Y and Z of the selected support tab. The Z-values can be edited for any tab, the X and Y values only for tabs that are oriented parallel to the X-axis or the Y-axis..

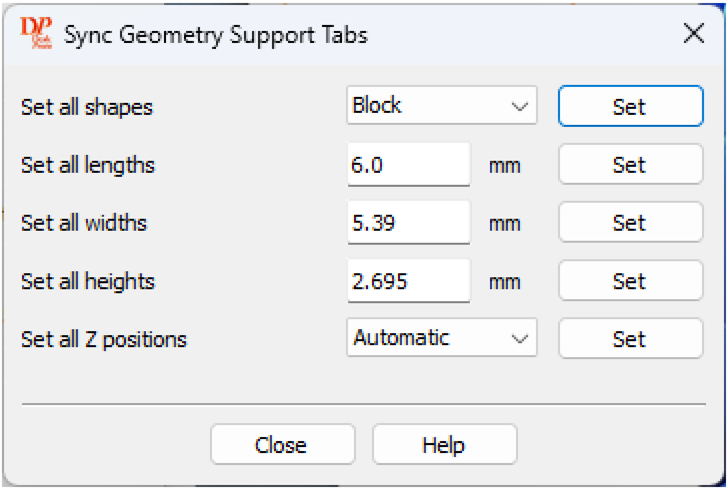
Finally the **Boundaries of the geometry** and the coordinates of the current **Mouse position** are displayed, to assist you when entering coordinate values for the support tab.
All coordinate values can be displayed either in CAD coordinates or, when your check **Display translated coordinates**, in NC file coordinates. The difference is the translation that has been set on the Translate tab of the Part parameters.

Notes:
It is possible to **add your own support shape**.
The base support tabs are present as STL files in files in C:\ProgramData\DeskProto 8.0\Supports\
You can add any STL file here, and use that 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 correct (internally DeskProto creates the support by copying, scaling and rotating the base support tab as defined in this file). This 1 x 1 x 1 mm is also true for inch users.

When using a DeskProto Script or a DeskProto Custom wizard it is possible to create support tabs that are rotated around X and/or around Y. For such tabs either editing is possible without the rotation options (when these rotations were multiples of 90 degrees), or editing is not possible at all. In the latter case you will not be able to open this dialog.

For software specialists:
you can set the **location of this supports folder** as follows:
For Windows change registry key HKCU\Software\Delft Spline Systems\DeskProto\8.0\Preferences\File Locations\SupportsLocation
For MacOS change line "8 0.Preferences.File Locations.SupportsLocation" in file DeskProto.plist
For Linux change line "Preferences\File%20Locations\SupportsLocation" in section "[8.0]" of file DeskProto.conf.
Only edit the registry or these files when you are qualified and know exactly what you are doing !

4.2.2.3 Sync Geometry Support Tabs



The Sync support tab dialog offers options to synchronize the geometric properties of all geometry support tabs.
You can reach this dialog by using the **Sync all** button on the Set Geomtry Support tabs dialog.

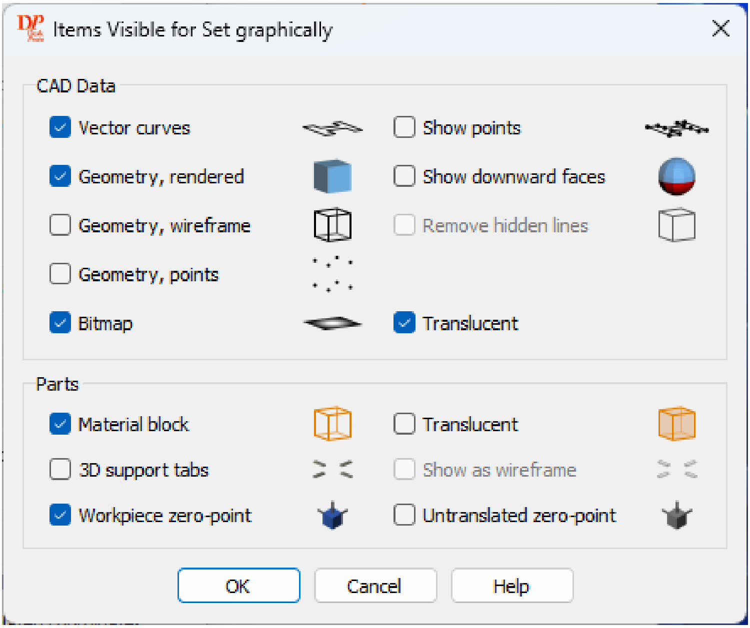
You can set the Shape, the Length, the Width, the Height and the Z-position of the tabs:
pressing the **Set** button will change all tabs to the value shown in this dialog.

The **Shape** of the tab can be set using a combo-box (button with a drop-down menu). As default five shapes are available: Block, Cone, Cylinder, Pyramid and Wedge. The Cone, the Pyramid and the Wedge are asymmetric: one end is much smaller than the other end. The tab will be connected to the geometry with its smallest side, making it easy to finish the part after removing the support tabs.

The **Length**, the **Width** and the **Height** are the dimensions of the tab - at least for a block-shaped support tab. For the other shapes these are the dimensions of the bounding box. When changing the dimensions of the selected block, the center point of the block will remain the same.

The **Z-position** of the block cannot be seen in the top view that this dialog shows as a default: change it to front view or side view to see the-position. Here as well a combo-box button, offering the options Top, Centre, Bottom and Automatic.

4.2.2.4 Items Visible for Set Graphically

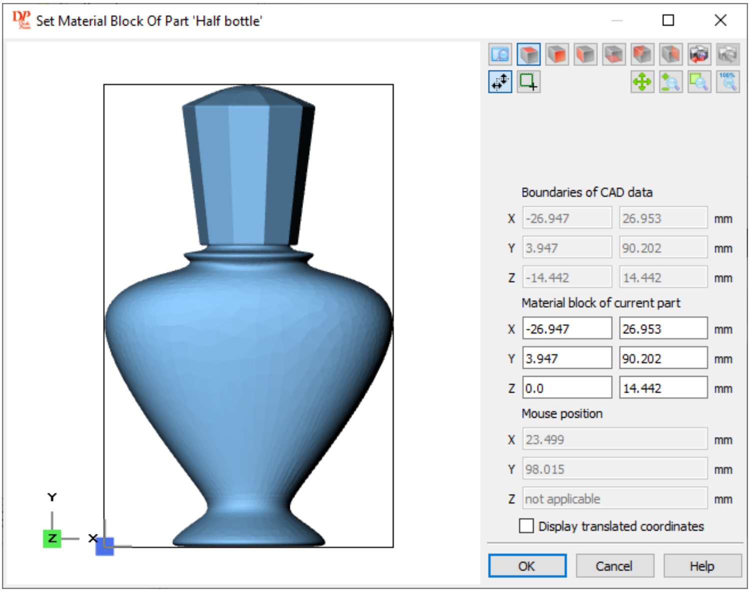


In DeskProto a number of dialogs are present that show a drawing of the part in order to **graphically** set a parameter:
Set Material block (Part parameters)
Set Support tab (Part parameters)
Set Center (Operation parameters, for strategies Circular and Radial)
Set Area (Operation parameters)
Set Freeform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

When graphically setting a 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 on any of the options in this dialog see the Help page of the Items Visible dialog for the main screen.

4.2.2.5 Set Material Block / Area



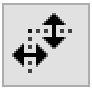
This dialog is used for two settings in DeskProto, each with a different name. The contents of the dialog is identical though.
The dialog **Set Material Block** makes it possible to graphically set the size and position of the material block. You can reach this dialog by using button 'Set Graphically'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

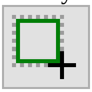
The dialog shows a new drawing of your part, with a rectangle that indicates the block or the area that you are editing. You can edit the rectangle by 'picking' one of the sides with your mouse and then moving it (as default the button 'Adjust boundary' now is active). Only the six main views can be displayed, as only then the block to be edited is drawn as a rectangle.

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 functions of this dialog: graphically setting the block or area. Note that these are “mouse buttons” as well: of the five mouse buttons only one is active at any time.

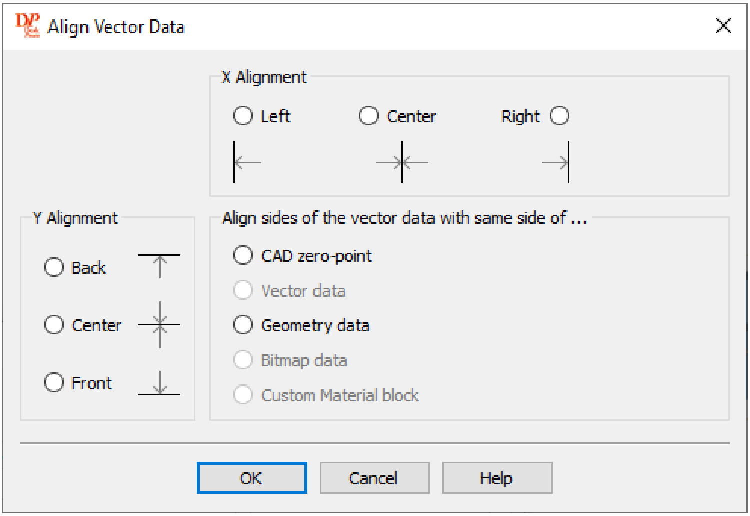
 This button sets the mouse function to **Resize** the current block or area by dragging each of the four sides to a new position.

 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.

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.

4.2.2.6 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 on tab page Transform, for Bitmap on tab page 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 (so aligning in a top view), panning along Z and rotating the data 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.
- **Vector data**. The bounding box of all vector curves. Only available for bitmap data, in case one or more vector files have been loaded as well.
- **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 as well.
- **Material block**. Only available in case a custom block has been defined.

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

4.2.3 Operation

4.2.3.1 Vector Operation Parameters

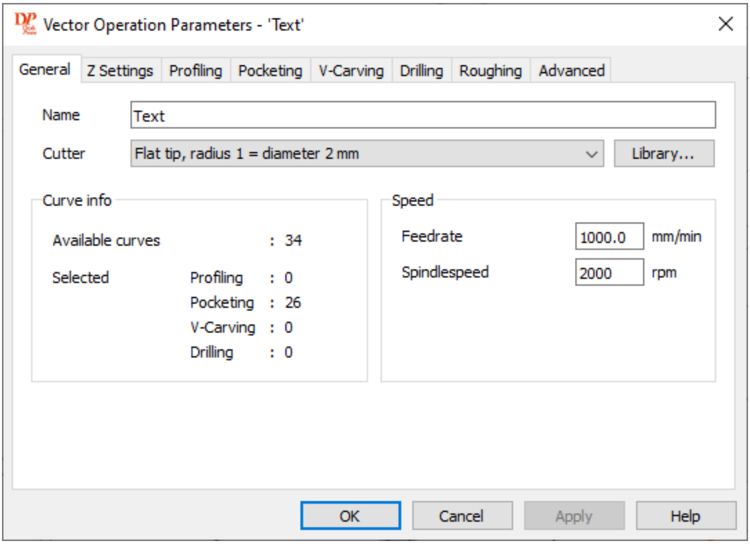
DeskProto features three different types of Operations: this dialog is for the Vector Operation - in addition also dialogs for a Geometry Operation and for a Bitmap Operation are available.

The Vector Operation parameters are divided into 8 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 a vector operation line in the project tree (one of the third level items).
Or right-click on a vector operation line 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
Here 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, normally it is not used in the NC program file.
Two exceptions: you can write the name in the NC file (in a comment line) using a postprocessor placeholder, and in case the NC output is in more than one file (for instance due to a cutter change) DeskProto will add operation names to the NC file names.

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 in the Cutter library (button **Library**).

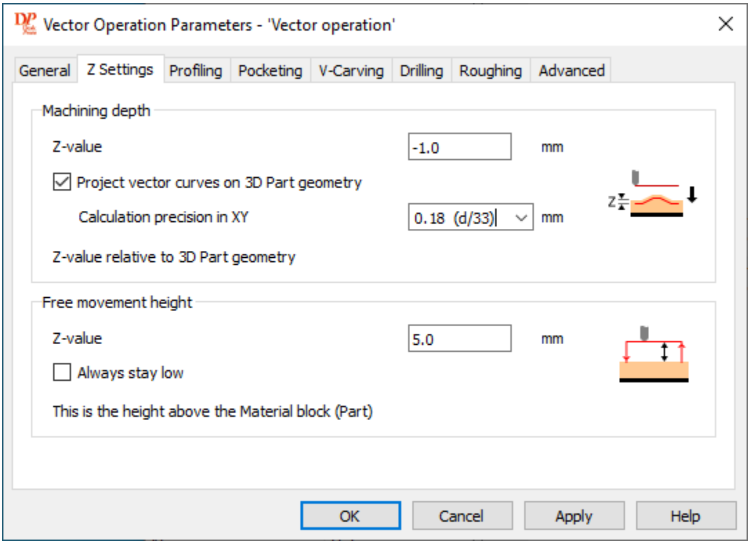
Curve info
The Vector data that has been loaded consist of a number of vector curves. In order to use curves for toolpath calculation they need to be selected first: for Profiling, for Pocketing, V-Carving and/or for Drilling. It is possible to combine two or more of these four toolpaths types in one 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 four tab pages just mentioned.

Speeds

The Speed settings are identical to the settings offered in the Geometry Operation Parameters: see the explanation on **Feedrate** and **Spindle Speed** there.
An option to reduce the feedrate for plunge movements can be found on the Advanced tab.

Laser
When you have selected a laser cutter one more group of settings is present. The **S-value** (which replaces the spindlespeed) sets the power of the laser. The range is set in the Machine definition, it can for instance be a percentage (range 0 - 100).
The **Pass count** is used when your laser is too weak (power too low) to achieve the burning that you need: you then simply can make it follow the same toolpath more than once.

Z settings parameters



The **Machining depth** defines the Z-coordinate for the actual toolpath. You can describe this as the *pen-down level* when plotting a 2D vector drawing. The depth is measured from the top of the block. This means that a positive value does not make sense here. Two exceptions are present:

- when you have checked the option 'Use Z-values' (in the Project parameters) the machining depth is measured from the Z-value of the curve
- when you have checked the option 'Project vector curves on 3D part geometry' (see below) the machining depth is measured from to the height of the geometry.

This will be shown in the dialog: the screenshot above shows the second exception.

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 constant 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.
When the vector data contains Z-values **and** in the Project parameters the option "Use Z-values" has been checked, the use of this option "Project vector curves on 3D part geometry" will make DeskProto ignore the Z-values in the CAD-data.

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 Z-values of 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.

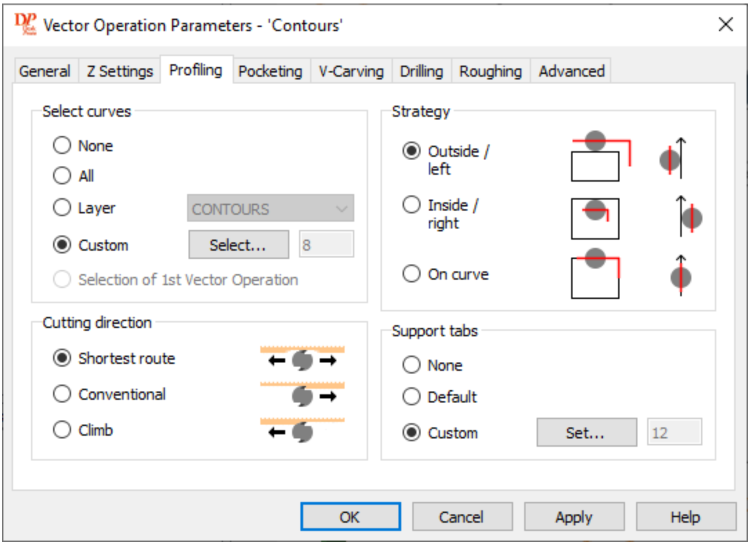
The **Free movement height** specifies the height at which the cutter can freely move over the block, so without touching material. This is also called the Safe height. As in the analogy with a pen plotter the machining level is the *pen-down level*, the free movement height can be seen as the *pen-up 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 not 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.

A special feature will be active when you select a laser cutter. For laser engraving the laser has to be switched off during positioning movements. DeskProto will do this automatically, using the commands as defined in the postprocessor.

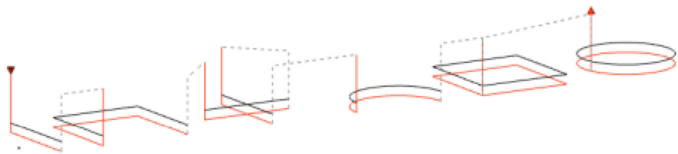
When the option **Always stay low** is checked, all positioning moves are done at normal feedrate. When only Vector data is present the Z-level of these positioning movements will not change: the specified height above the top of the block. When a Geometry has been loaded as well the positioning movement 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.
This does not apply to laser cutters: then any positioning movement the laser needs to be switched off..

Profiling parameters



Four different toolpath types can be generated based on the vector curves: Profiling, Pocketing, V-Carving and Drilling. This tab page contains the settings for **Profiling**: the 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 options **None** and **All** will be clear.
The option **Layer** refers to the layers in the DXF drawing (or rather the CAD file), and provides an easy way to select all curves in a certain layer. Which is convenient as in many design files all curves to be done by the same cutter are combined in one layer. In case for a curve no layer information is present in the CAD-file DeskProto will assign the curve to a layer called "(DP-default)".
To make a **Custom** selection it is needed to open dialog Edit curve selection using button **Select...**.
The fifth 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), five open curves (the + consists of two curves) and two closed curves (on the right side). Selecting **All** for this drawing will result in 7 curves being selected here: all except the single point.

Three **Strategy** options are present, setting how 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 CAD file).
For closed curves inside this curve (so for nested curves) the cutter will travel on the inside of the curve, creating holes in the part. The next nesting level again will be with toolpath on the outside, and so on. DeskProto will start machining the innermost curves and than work it's way to the outside, in order to keep the part clamped when machining.

Inside/right is also 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 CAD file).
For closed curves inside this curve (so nested curves) the cutter will travel on the outside of the curve, creating islands within the hole. The next nesting level again will be with toolpath on the inside, and so on. DeskProto will start machining the innermost curves and than work 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 pen 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 trimming thermoformed parts. The toolpath will copy the Z-value of each point in the curve and subtract the machining depth. For strategies Outside/left and Inside/right the Z-value of the closest point in the curve will be used.
Take care: when curves and/or toolpaths at different heights share the same XY position the algorithm may use an incorrect Z-value, so please check your toolpaths when using this option for a 3D vector curve.

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 that makes 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

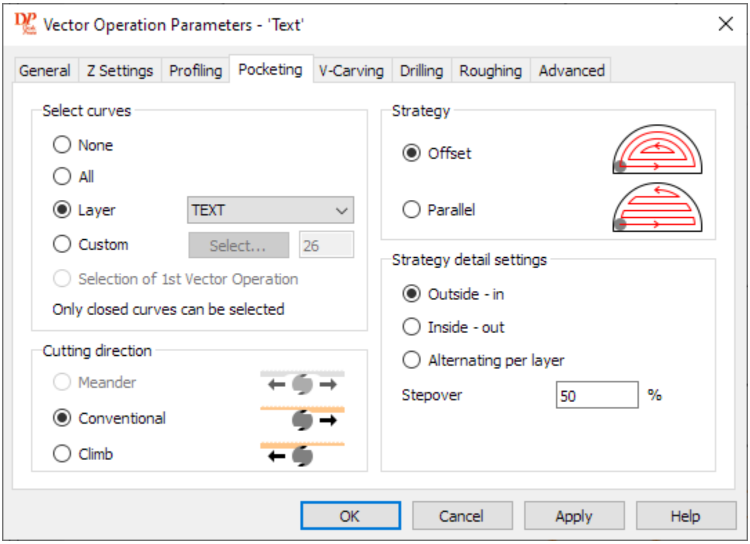
Support tabs can be used to keep the part connected to the rest of the block. When cutting a closed profile at full depth in sheet material a separated part will result: the material inside the profile is no longer is connected to the rest of the sheet, and the cutter may send it flying away. Support tabs are *interruptions in the toolpath*, acting as connection bridges to hold the part on its place. This allows you to clamp the material 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 Vector Support tabs settings.
Custom allows you to exactly define the location and size of each tab, button **Set** opens the Set Vector Support tabs dialog.
The edit box on the right will show the number of tabs, for all three options.

Vector support tabs are available only for vector curves without Z-values (2D curves). So they also cannot be used when a 2D curves is projected on a 3D geometry, or when for a 2D vector file the option "Use Z-values" has been checked in the project parameters.

Pocketing parameters



Four different toolpath types can be generated based on the vector curves: Profiling, Pocketing, V-Carving and 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 options **None** and **All** will be clear.
The option **Layer** refers to the layers in the DXF drawing (or rather the CAD file), and provides an easy way to select all curves in a certain layer. Which is convenient as in many design files all curves to be done by the same cutter are combined in one layer. In case for a curve no layer information is present in the CAD-file DeskProto will assign the curve to a layer called "(DP-default)".
To make a **Custom** selection it is needed to open dialog Edit curve selection using button **Select...**.
The fifth 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), five open curves (the + consist of two curves) 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.
When a curve that seems closed is in fact open you can find the error by checking "Show points" for Vector curves in the Items Visible dialog. The start point of the curve will be indicated by a larger dot. Closed curves do not have a starting point, so when you see a large dot it indicates a gap in the curve.

Two **Strategy** options are present to create a pocket:
The **Offset** strategy fills the pocket with toolpaths that follow the shape of the outside curve (the contour that defines the pocket).
The **Parallel** strategy fills the pocket with toolpaths parallel to the X-axis.
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 **Offset:**

For the start and the end of the toolpath you can select one of three options:
- Offset toolpaths normally start with the outmost path and then progress to the middle: **Outside - in**,
- the reversed option is to start in the middle and progress to the outmost path: **Inside - out**,
- and the third option alternates these two per roughing layer, thus preventing positioning movements when several roughing layers are present: **Alternating per layer**.

The **Stepover** is the distance between two parallel toolpaths. It is set as percentage of the cutter diameter (flute 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. Using a flat cutter the pocket will have a flat bottom. With a ballnose cutter cusps will remain between the toolpaths. And with conical cutters the default Stepover value of 50% will be far too high - conical cutters are not optimal for pocketing anyway.

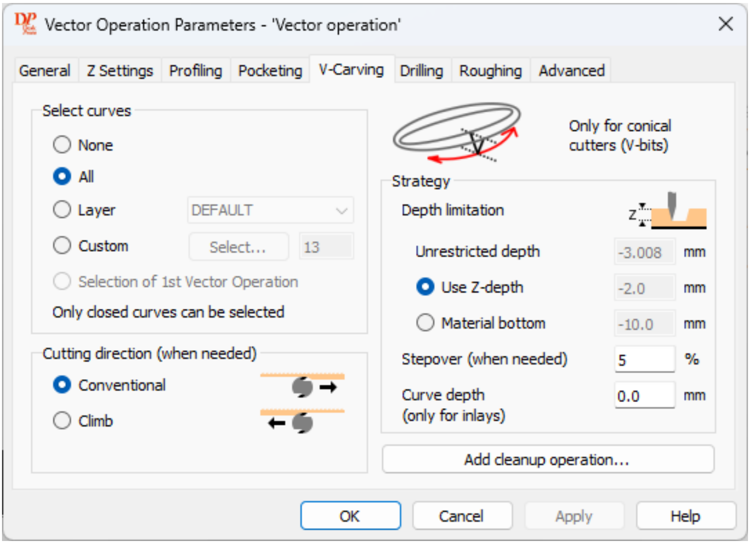
For **Parallel**:
Again you can select one of three options for the start and the end of the toolpath:
- Parallel toolpaths normally start minimum Y and continue to maximum Y: **Front to back**,
- the reversed option starts at the maximum Y: **Back to Front**,
- and the third option alternates these two per roughing layer, thus preventing positioning movements when several roughing layers are present: **Alternating per layer**.

The **Stepover** setting is the same for both strategies (see above), set as percentage of the cutter diameter (flute diameter).
For parallel paths a third setting is available:
Parallel toolpaths as said are parallel to the X-axis, the option **Angle with X-axis** allows you to change this.

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 actual cutting direction on the outer profile of the pocket will be the reverse.

For both strategies it is possible to add a finishing 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.

V-Carving parameters



Four different toolpath types can be generated based on the vector curves: Profiling, Pocketing, V-Carving and Drilling. This tab page contains the settings for V-Carving. As this is a completely different way of using the CNC milling machine, please read the V-Carving page before using this toolpath type.
The settings on this page are available only when a **conical cutter** ('V-cutter') has been selected for the operation, otherwise all settings will have been 'grayed out'.

Select curves allows you to specify which of the available vector curves need to be used to generate pocketing toolpaths.
The options **None** and **All** will be clear.
The option **Layer** refers to the layers in the DXF drawing (or rather the CAD file), and provides an easy way to select all curves in a certain layer. Which is convenient as in many design files all curves to be done by the same cutter are combined in one layer. In case for a curve no layer information is present in the CAD-file DeskProto will assign the curve to a layer called "(DP-default)".
To make a **Custom** selection it is needed to open dialog Edit curve selection using button **Select...**.
The fifth 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 V-Carving **only closed curves can be selected**, as an open curve does not define a pocket to be V-Carved.

When a curve that seems closed is in fact open you can find the error by checking "Show points" for Vector curves in the Items Visible dialog. The start point of the curve will be indicated by a larger dot. Closed curves do not have a starting point, so when you see a large dot it indicates a gap.

The **Cutting direction** applies only when a horizontal bottom surface is present in the V-groove, caused by the maximum depth that you have set. It applies to the toolpaths that machine this surface (which is in fact a pocket), not to the paths that machine the walls of the V-groove.

Two **Strategy detail settings** are available for V-Carving:

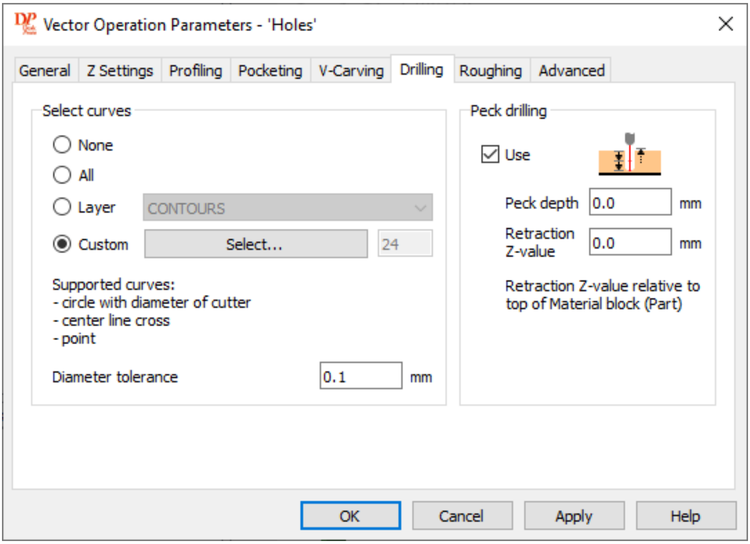
You can set a **Depth limitation** for the V-groove to be machined.
When V-Carving the depth of the groove is determined by the angle of the cutter and the width of the groove as defined in CAD. Field **Unrestricted depth** will show the maximum depth of the V-Carving toolpath for your settings. This information is available only after you have calculated the toolpaths.
The default depth limitation is **Use Z-depth**. This means that DeskProto will use the Machining depth (that you have set on tab Z-settings) as maximum depth. When that depth is less than the unrestricted depth the resulting groove will have a **flat bottom**.
The maximum depth limitation that you can set is the **Material bottom**, as making a V-groove that is deeper than the bottom of the block is not possible.

The **Stepover** is the distance between the toolpaths when machining a horizontal bottom surface. This applies only 'when needed', so only in case such horizontal bottom is present. The value that you enter is a percentage of the diameter of your cutter. Machining this bottom is in fact a pocketing operation, and conical cutters are not very suited for pocketing as they will not create a flat bottom surface. Not even with the low default Stepover value of 5%. You can make the bottom flat by adding a Cleanup operation, as described below.

The last setting in this group is the **Curve depth.** This is a setting that needs to be used **only** when you are creating the male part of an **Inlay**. It is a setting to position the curves to be followed by the cutter at a certain distance (in Z) below the top of the block. The inlay page explains in detail how this works.
For normal V-carving this setting should remain 0.0, meaning that the curves are positioned on the top surface of the block. Then the V-Carving toolpaths will create sharp corners on this top surface, where they will be visible as designed in the artwork.
Some other CAM programs call this Curve depth the 'Start depth'. DeskProto does not, as the roughing layers for curves at a certain curve depth will start at the top of the block, so that is in fact the Start depth.

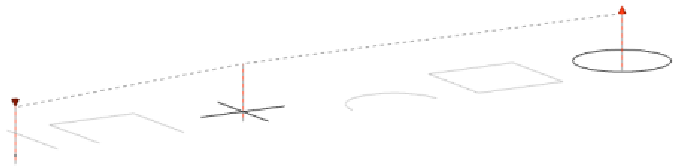
Finally you will see that an extra button is present on the V-Carving tab: **Add cleanup operation...**. This button opens a dialog that allows you to add an extra operation to this part: a 'cleanup operation'. This is meant for V-Carving operations where the V-Carve has a flat bottom, so the button is activated only which such flat bottom can be present: with a restricted Z-depth.
The V-cutter that is used cannot machine a flat bottom surface, as the V ends in a sharp tip. The cleanup operation will use a flat endmill and pocketing toolpaths to clean it up: make this bottom surface flat. DeskProto will automatically enter the correct settings in the pocketing operation that will be created.
When you have added a cleanup operation the text on the button will change to **Update cleanup operation**, and so will its function. The Update button will be activated when a relevant setting in the V-Carving operation has changed.

Drilling parameters



Four different toolpath types can be generated based on the vector curves: Profiling, Pocketing, V-Carving and Drilling. This tab page contains the settings for **Drilling**: the cutter will move along the Z-axis and drills a hole in the block, at the XY location as defined in the drawing, and a 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 drilling toolpaths.
The options **None** and **All** will be clear.
The option **Layer** refers to the layers in the DXF drawing (or rather the CAD file), and provides an easy way to select all curves in a certain layer. Which is convenient as in many design files all curves to be done by the same cutter are combined in one layer. In case for a curve no layer information is present in the CAD-file DeskProto will assign the curve to a layer called "(DP-default)".
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 and the lines intersect exactly in the middle.
- circles with a diameter that equals the cutter diameter

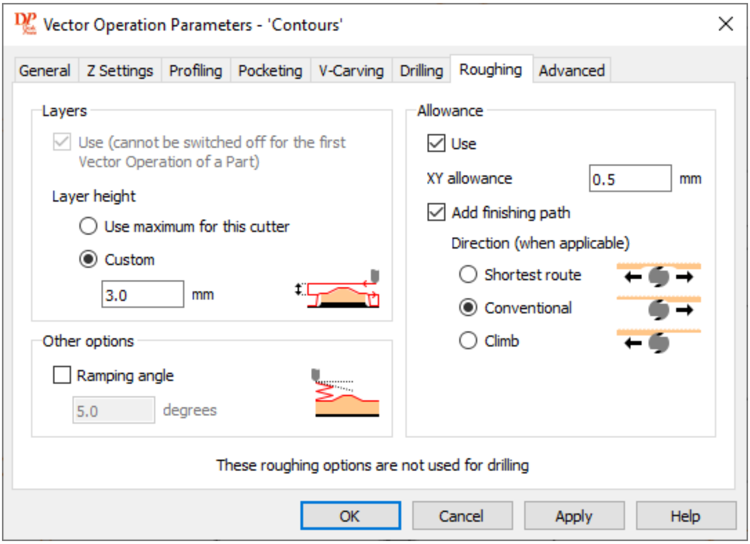
The drawing above contains 8 vector curves: one single point (on the left side), five open curves (the + consist of two curves) and two closed curves (on the right side). Selecting **All** for this drawing will result in 3 curves being selected here - that is only in case the diameter of the cutter is correct for both the + and for the circle. For a cutter with any other diameter only one curve will 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 or drill 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. You will need to enter a (positive) value here: the default value of 0.0 is not permitted.
The **Retraction Z-value** sets the height (above the top of the block) that will be reached in the retraction movement.

Roughing parameters

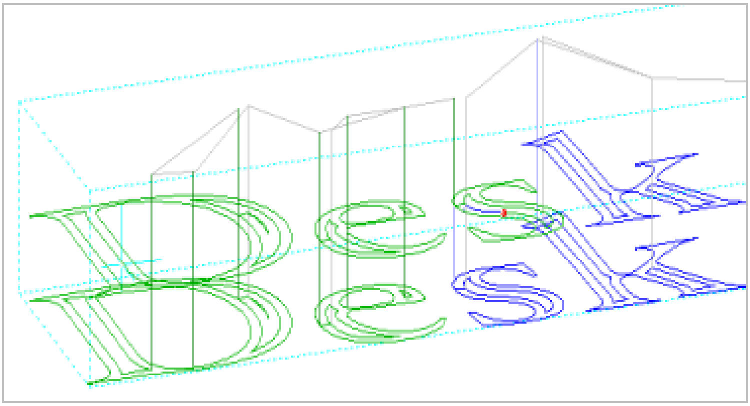


Roughing is applied when the cutter cannot reach the required Machining depth in one cutting movement, either because the cutter's cutting length is not sufficient or because the material is too hard to machine all at once. The total depth then will be reached in a number of cutting passes (layers), 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, leaving only a thin allowance, the load for the finishing toolpaths will be low which will reflect on the resulting quality (less vibrations in the cutter).
Roughing is not possible in case only curves for Drilling has been selected.

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 (dive deeper) than is possible for the cutter: its **maximum layering height**, as explained on the Geometry roughing page. 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 Maximum for the cutter or to a Custom value. The custom value may not be higher than this maximum. The default layer height equals the **Maximum** layering height, which for flat cutters equals its cutting length bur for ballnose cutter is a bit less (as explained on the Geometry roughing page). 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 you can 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 (unless you check the option 'Complete all layers per island').
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 will be 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 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 (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.

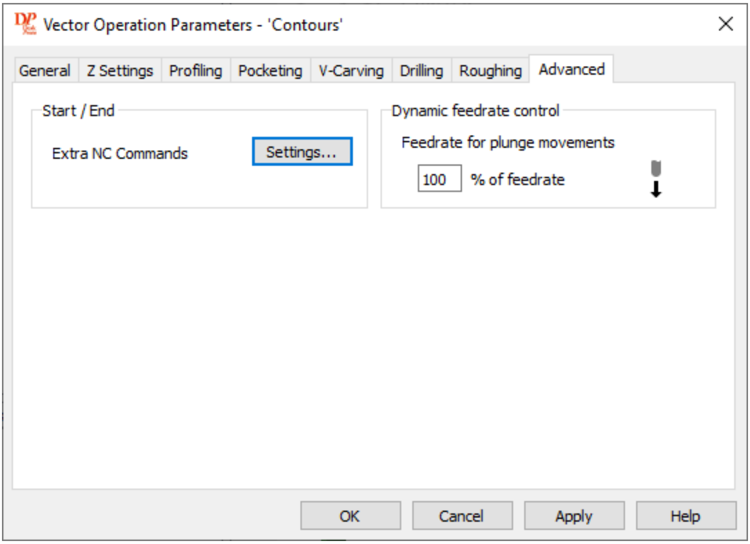
Entering a Ramping angle makes DeskProto replace this vertical movement by a series of ramping movements: go down along a sloping line. See the small picture in the dialog for this option. You can set how steep this sloping line needs to be by entering an angle value in degrees: this is the angle between the sloping line and a horizontal line.
The ramping angle and the length of the vertical movement determine the length of the horizontal component of the ramp. When this horizontal component is less than 10% of the cutter radius DeskProto will not apply ramping.

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 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. When Pocketing, this finishing path will be a Profiling path (as the allowance is XY).

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). This setting does not apply for V-Carving toolpaths.

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, or to add comments using the Postprocessor placeholders. Button **Settings...** will open the Operation Start/End settings dialog.

The **Feedrate for plunge-movements** makes it 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.

4.2.3.2 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.

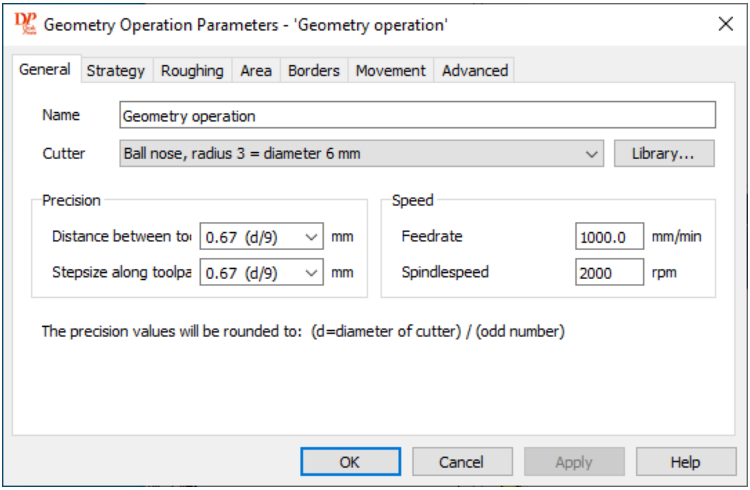
This dialog can be reached via the Parameters menu, third option.
Or you can double-click on a geometry operation line in the project tree (one of the third level items).
Or right-click on a geometry operation line and select Operation Parameters in the context-menu.

When a geometry operation is part of a Chain with more than one geometry operation, an extra button will be present: **Apply to chain**. After changing one of the operation's parameters, for instance the Strategy, you can press that button in order to apply this change to all geometry operations in the chain.
Note: this button is present **only** in geometry operations, and it will not affect any bitmap operations and/or vector operations in the chain.

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
Here 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, normally it is not used in the NC program file.
Two exceptions: you can write the name in the NC file (in a comment line) using a postprocessor placeholder, and in case the NC output is in more than one file (for instance due to a cutter change) DeskProto will add operation names to the NC file names.

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 cutter later, for some detailed areas.
Adding or removing cutters to/from the list can be done in the Cutter library (button **Library**), where you can also view and 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). In most cases both distances can be set on the same value. The smaller these two 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 in fact 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, this is not possible: 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. Which is compensated by its advantages such as calculation speed and robustness.

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 in the pull-down menu match this formula. You may enter a custom value as well, which in case it does not match the formula will be rounded to the first smaller value that matches. The reason to use this formula is that the resulting (physical) part will be more accurate, due to the grid-based algorithm used by DeskProto for its calculations. Both the geometry and the cutter are represented by a grid with the same grid-cell size, and then the cutter diameter exactly matches an odd number of grid-cells the results will be most accurate.

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 diameter or the tip diameter).
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 in this division. 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 deactivated (grayed out).
Same for other cutter types, though then adding the skin will change the geometry of the virtual cutter (sharp corners will be rounded).

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 setting the Feedrate in this dialog are set Preferences. The units for the Feedrate used in the NC file are set in the postprocessor of the machine that you selected for your part. If these settings are different DeskProto will convert the value that you enter. Whether or not you can use decimal values (numbers behind the decimal point) for the Feedrate 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. Very useful are the options is to let DeskProto automatically reduce the Feedrate in high chipload conditions or for plunge movements, 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 (or rounds 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 that is needed to get the same actual cutting speed of the tool's cutting edge.
Some of the machines that offer a laser engraving option use the Spindlespeed value to set the laser power when the laser option is active.

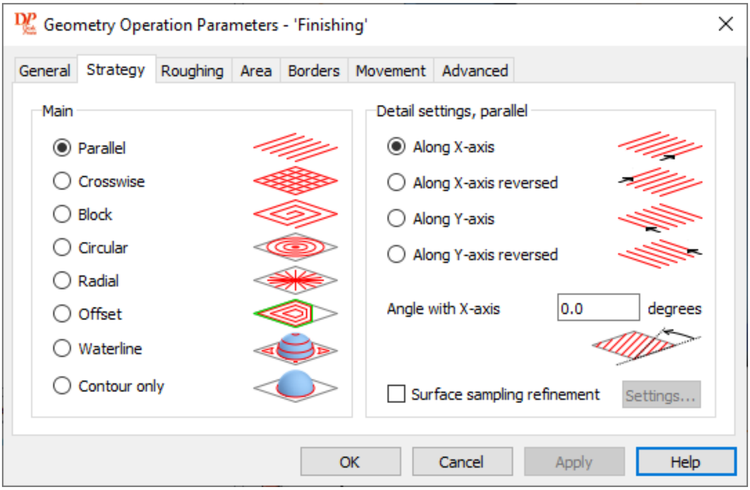
Note:

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



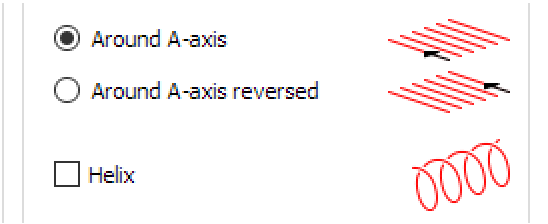
The eight available **Main** strategies should be clear from the pictures drawn in the dialog. Each main strategy (shown at the left) has its own **Detail settings** at the right.

Parallel

Parallel toolpaths is the default strategy: DeskProto will project a series of parallel toolpaths over 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 with a constant Y position), 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**), respectively 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: that 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:

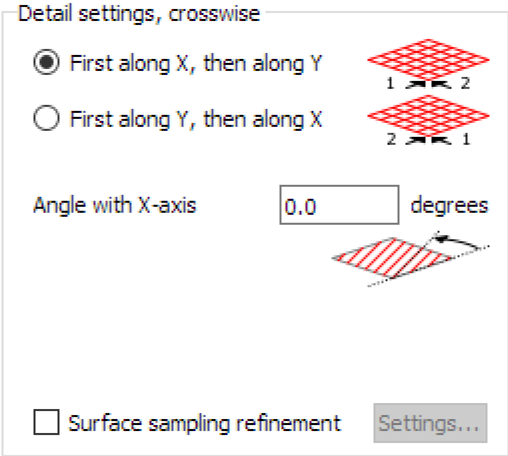
- in the Rotary Machine settings option "A-values may exceed 360" has been checked
- on tab page Movement either Conventional or Climb has been set as Cutting direction
- the Parallel toolpath detail settings are either Around A or Around A reversed
- The Area to be machined includes the complete cylinder (so range from 0.0 to 360.0 for A)
- you have a Geometry operation: the Helix is not supported in Bitmap operations.

When this option is **not** 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. Checking 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 many more rotary parts.

Using the Helix option combined with Roughing layers (and sorting) will not give good results: the Helix path will be interrupted when skipping what is done in previous layers. Best is to use Helix only for finishing.
One more remark about the helix toolpath is that rotary positions on your machine need to be exact. When a 360 degree movement on your machine actually is 359.9 degrees, this difference will be invisible. However, after one hundred rotations (so at A=36000 degrees) the error will be 10 degrees: clearly visible. The result will be a twisted part, and finishing won't match roughing.

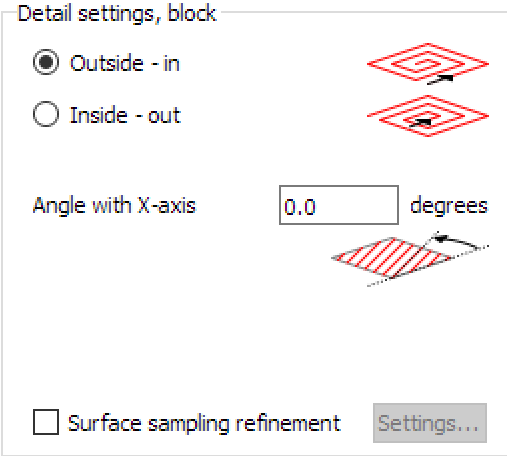
The last detail setting for Parallel is the **Surface sampling refinement**: which will be explained below.

Crosswise



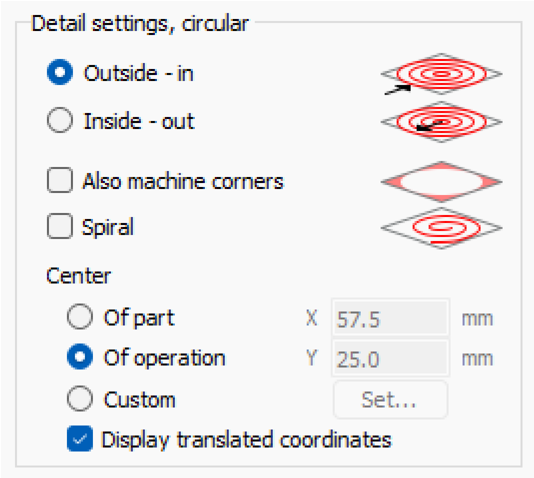
Second strategy is Crosswise. This is the same as creating two operations where one of them uses parallel to X and the other one uses parallel to Y. When you would use two operations there would be redundant calculations though, resulting in longer calculation time: hence this strategy. This option is useful in case the model you want to produce must have a very high 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.

Block



The Block strategy combines toolpaths parallel to X and Y to a sort of rectangular 'spiral'. These are probably the most efficient toolpaths 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.

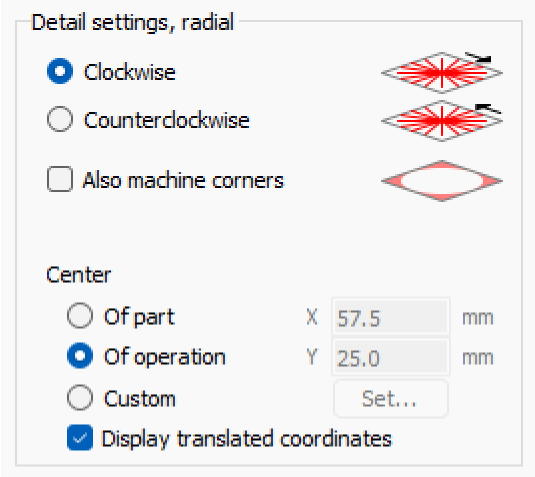
Circular



Circular is a completely different strategy: it does not use the rectangular base pattern (grid) that is applied in the above strategies. 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 is very well be suited for round geometries, like rings or cups.

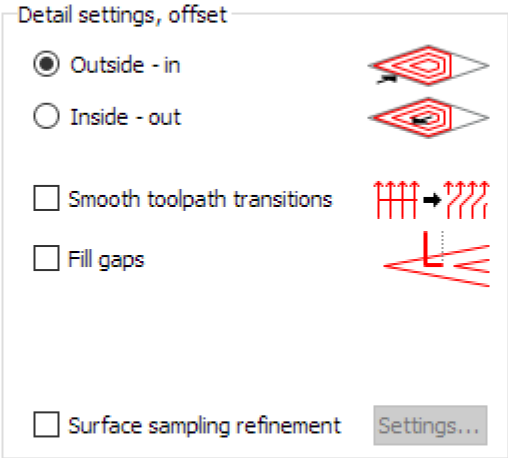
Detail settings are:
Inside out versus **outside in** (same as for block strategy)
Yes or no **machine the corners**: this concerns the area that is inside the rectangular area to be machined but outside the largest circle that touches all four sides of this area. This option is available only in case 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 value. 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 display and 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 the 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.
Strategy Circular is not available for rotation axis machining.

Radial



Radial is the complement of circular: same Z-grid, however now radial toolpaths (so perpendicular to the circular paths). The same Detail settings apply here as well, except for the Spiral. And the sequence of the toolpaths now is called **Clockwise** versus **Counterclockwise**.
As milling direction now Meander is advised, as Climb and Conventional will need a positioning movement after each path.
Strategy Radial is not available for rotation axis machining.

Offset

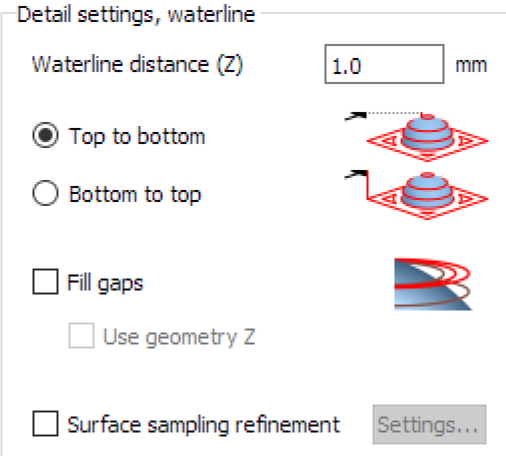


Offset machining generates toolpaths as offset lines to the contour round the area to be machined.
This area can be defined using option Skip Ambient, using a (freeform) Area to be machined, or using a combination of both. For a rectangular custom area the paths will be similar to the Block strategy, 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 set a Freeform area and then automatically generated a contour that follows the outline of the geometry (in a top view). 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 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 this prescribed distance, creating gaps in the toolpath pattern. See the small drawing in the dialog. 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

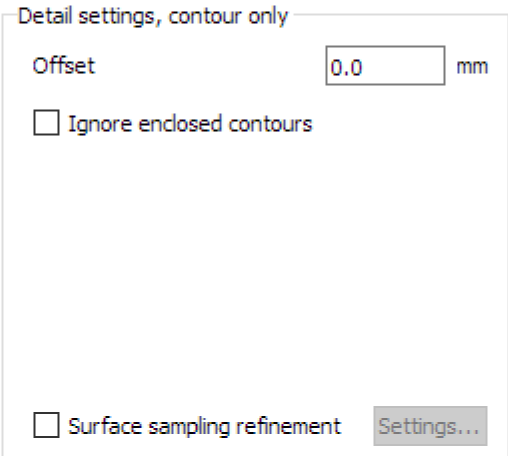


Waterline machining produces toolpaths on a constant Z-level (just like the waterlines over a ship's hull). Such strategy is also called contour machining or Z-plane machining. While the difference with toolpaths on constant X or on constant Y seems small, in reality the difference is huge as a completely different calculation algorithm is needed. In the Detail settings an extra parameter is present: 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 used when horizontal surfaces (gaps) have to be filled.

Important is to realize that only on each prescribed height level 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 gaps** is present. 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: a gap. This gap may even be larger than the diameter of the cutter: resulting in islands of material remaining after completing that operation. The option Fill gaps 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 will have the same Z-value, so a visible staircase effect will be the result.
The sub-option **Use geometry Z** reduces this staircase effect by calculating Z-values for all toolpaths that are created to fill the gap.
The **Surface sampling refinement** will be explained below.

Contour only

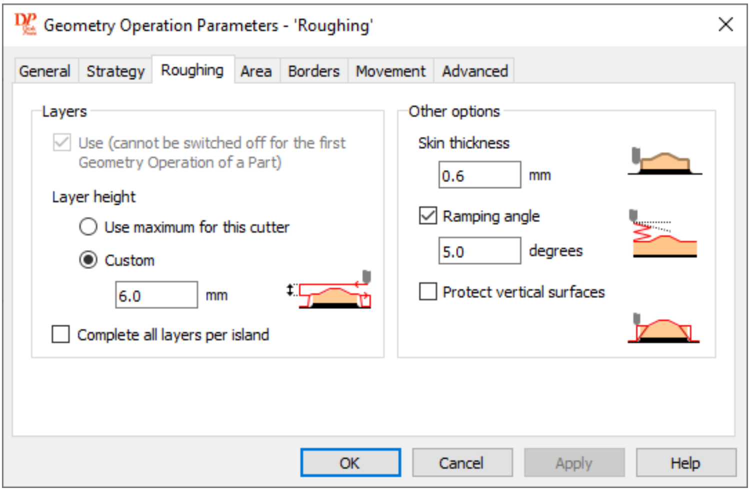


The last strategy, Contour only, is in fact an additional strategy as it does not machine the complete part: only the outline of the geometry (outer contour) at ambient level is machined. This can be used after some other strategy: to smoothen the model (when DeskProto creates toolpaths that are parallel 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 normal result for this strategy will include a contour path within this hole (nested contours). Checking this box will make DeskProto create a toolpath only for the outer contour.

Surface sampling refinement
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.

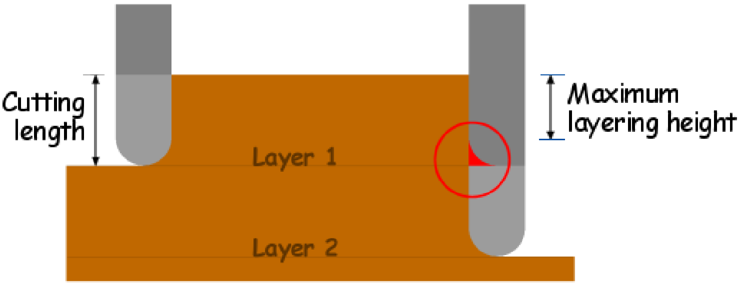
Roughing parameters



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 that 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 use two operations, by adding one in the General Part parameters.

Layers

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. Layers are also known as roughing passes. The default layer height equals the complete **maximum layering height** of the cutter, see the image below. 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 image above shows why for **ballnose cutters** the Cutting length of the cutter cannot be used as maximum for the layer height. When machining layer 1 rest-material will remain at the ballnose top of the cutter, which then in layer 2 would collide with the shaft of the cutter. In the image this collison is shown in red. As the shaft of the cutter cannot cut this is not permitted. So DeskProto uses a **maximum layering height** that is smaller than the cutting length of this cutter. When this max layer height is smaller than the radius of the cutter (and when roughing layers are applied) DeskProto will show a warning that "*the selected cutter may not be suitable for roughing layers*".

For **conical cutters** the situation may be worse, as for such cutters often only the conical section can cut (so no vertical cutting edge as in the above image). For such cutters DeskProto will show the above warning when applied in a roughing operation: the user then can decide whether or not this is acceptable. Many jewelers for instance use only conical cutters ("V-bits"), however for machining wax models this bit of rest material is no problem at all. The warning is optional, so a jeweler can simply uncheck the "Always show" checkbox for this warning.

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.

When you absolutely need a first operation without layers, and you are very sure that sufficient material has already been removed to do this, you can add an extra Geometry Operation to the part, before the current operation, and make it invisible (an unused "dummy" first operation).
Note:
The first layer starts at the top of the material block. When your actual block is higher than the part you can define a custom Material block in DeskProto, having a higher Maximum Z value, to match the real block.

The option **Complete all layers per island** gives an alternative to the standard sequence of machining layer after layer. As default DeskProto will completely machine each layer before starting with the next layer. In some cases that is not what you want: when the cutter machines on various locations that are completely separated (for instance more than one geometry or more than one freeform area). Each location then can be seen as an 'island', as a positioning movement is required to travel to the next island.
DeskProto then will need to perform many such positioning moves, which makes the machining time longer than needed. Checking this option will make DeskProto machine island per island: so it will complete all layers of an island before moving to the next island.

The island status is determined using a rectangular XYZ bounding box round the toolpaths. So when for two actual islands the two bounding boxes overlap DeskProto will not recognize their island status.

Other Options

Entering a **Skin Thickness** results in a model that 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 when finishing the tool only needs to 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 immediately, so it will try to take off the skin of the complete vertical wall in one go. In case the wall is higher than the cutting

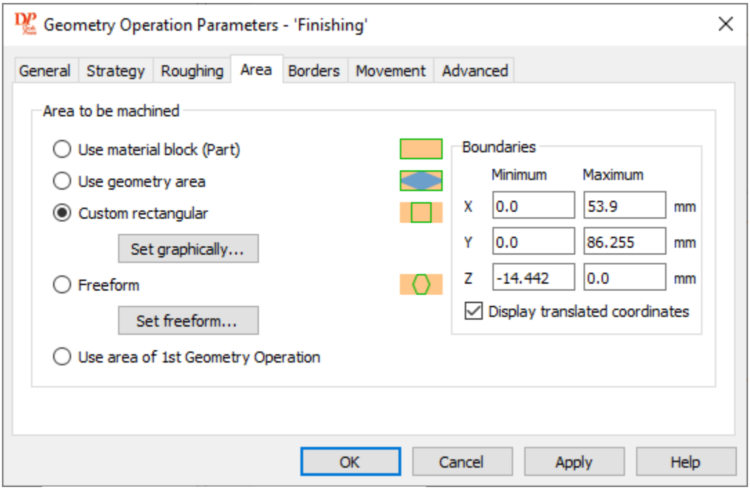
length of the tool this is a problem - for which no automatic solution is available yet. An easy workaround to solve it is to add an operation using the waterline strategy (between roughing and finishing), milling from top to bottom.

The **Ramping angle** is used when starting to machine. The cutter will 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 (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.
The ramping angle and the length of the vertical movement determine the length of the horizontal component of the ramp. When this horizontal component is less than 10% of the cutter radius DeskProto will not apply ramping.

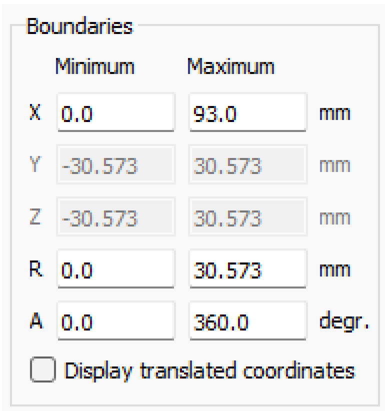
The option to **Protect Vertical Surfaces** is useful when roughing with a large value for the Stepsize along toolpath. In case the steps are large they may not “see” all geometry in-between and remove too much material. This may happen in case of vertical or steep surfaces in the geometry. Checking 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 replaced by staircases*, so only check this option in case needed.

A more advanced option to protect the Vertical surfaces (with a more extensive explanation) is present on tab Advanced. There you can set how steep the surface needs to be to activate this protection, by setting a height/step ratio. In this roughing option the height/step ratio is 1, which means 45 degrees. When this Roughing option has been checked the Advanced option will be disabled, as that would be overruled by the Roughing option.

Area parameters



Rotary machining:



The **Area to be machined** offers exactly what its name tells you: it limits the toolpaths for this Operation to cover only this area. It can be used for instance for a small area of the part that is very detailed and needs to be machined with a smaller tool in an extra operation. The 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 image above on the left shows six edit boxes. This is how the page looks in a part that uses three-axis machining. The area definition also applies to the Z-axis, allowing you to set a maximum machining depth and height for

the area. That can be handy: when your part has a flat top that does not need machining you can set the Max Z for the area a bit (say 0.1 mm) lower than the Z of this top surface: it then will fall outside the area to be machined.
For a part that uses **rotation axis machining** (4th axis) the page will show ten edit boxes (image on the right), more about that below.

The default option here is **Use material block (Part)**. This means that the cutter will machine the complete block, as defined in the Part parameters.

When your material block is smaller or larger than the CAD data that you loaded 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 (when rotary machining this is a cylinder). 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.

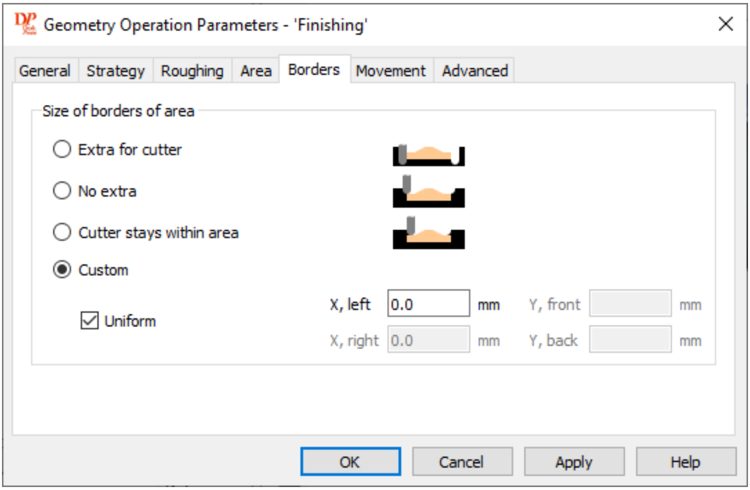
The second type is the **Freeform** area. This option allows the use of areas that are not rectangular, for instance a circle or any freeform contour (circular resp freeformed in top view). Button **Set freeform** pops up the Set freeform area dialog.
For a freeform area the Min and Max X and Y values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

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.

As said above, for **rotation axis machining** the options on this tab page are a bit different:
- an extra option is available: **Use material block above rotation axis.** Which is many cases is all you need, as the area below the rotation axis will be machined after rotating 180 degrees further.
- extra edit boxes are present for the min and max Radius and for the min and max Angle. Only six of these can be edited: min and max Y and Z can only be viewed. In order to make the cutter also **machine below the rotation axis** you need to enter a negative value for the minimum R (radius).
When the minimum R is higher than zero the resulting area will have a doughnut shape

Borders parameters



In order to machine all geometry in the area to be machined, the cutter needs to move a small distance outside this area. This is needed as otherwise the cutter cannot reach any (almost) vertical surfaces at the outside edge. The small images in the dialog illustrate why this is needed. In DeskProto this extra area called the **Border area**, and on this tab-page 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 (for an area that matches the part's bounding box).
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.

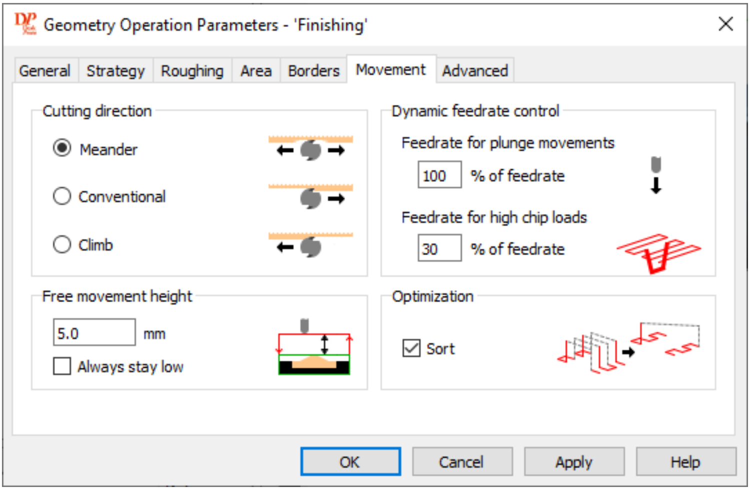
The second option, **No extra**, keeps the cutter positions (for the centerpoint of the cutter) exactly inside the area to be machined. Using this option it is still possible that the tool cuts away material that lies just outside the area, as half of the cutter reaches outside. DeskProto will anyway 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.

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. Four boxes are available only if the area to be machined is a rectangle, for a freeform area only two border sizes can be set: one for X and one for Y. And for rotary machining only the two X-borders can be set.

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. For **rotary toolpaths** cutting direction Meander results in a first movement from 0 to 360 degrees, next from 360 to 0, etc.

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 the cutter along the toolpath and the rotation direction of the cutter (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 the surface of your part, 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, as 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, also called the "Safe height". 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 values are allowed: in case of a negative free movement height the model and/or the cutter could be damaged.
When a Skin has been set (Roughing) the free movement level will be the Skin-thickness higher

This Free movement height (ZFree) value is 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 higher) 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 in that movement. 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.

Positioning movements:
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 - DeskProto tries if type 1 can be used, if not possible try type 2, if not possible use type 3:

1. Direct movement.
The cutter moves in a straight line between startpoint and endpoint of the positioning movement.
This happens when:

- the distance is **less than 1.9 times the Toolpath distance** ánd no geometry with a higher Z-level is present on the path.
As toolpath distance is used (also for type 2 as described below): for Geometry, Bitmap and projected Vector the Precision value, for normal Vector the Pocketing Stepover distance (or its default, being 50% of the diameter of the cutter tip).
or when:
- the distance is **less than the cutter-tip-diameter** ánd the toolpath is at constant Z ánd no geometry with a higher Z-level is present on the path.
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.

2. Lowered Z-height.
The cutter moves in a horizontal line, at normal Feedrate. The Z-height is Z-Free mm/inch above the highest point of the geometry below this line, including startpoint and endpoint (this does not work for a bitmap relief).
This happens when:
- the distance is **less than 1.9 times the Toolpath distance** (so when in type 1 geometry with a higher Z-level was detected on the path) ánd the cutting direction is Meander.
or:
- the distance is **less than the cutter-shaft-diameter** ánd the cutting direction is Meander.
or:
- the distance is **less than 10 times the smallest of** Toolpath distance, Cutter tip diameter ánd the cutting direction is Meander.
or:
- in the Operation parameters the **option "Always stay low" has been checked** ánd the cutting direction is Meander.
Exception: "Always stay low" is not (yet) possible when roughing layers are applied, as then for some strategies the lowered positioning movement might travel through un-machined material.
For Vector operations without geometry checking Always stay low will only change the Feedrate, not the Z-height.
The condition "Cutting direction is Meander" does not apply for Vector toolpaths.

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 travels 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.

Positioning movements for special cutters:
For the Special cutters that you can use in DeskProto these rules for positioning movements cannot be used. Two differences are present:
- **saw cutters** (saw-blade, chain-saw) can stay low in one direction but not in the other: in the other direction even the smallest positioning movement requires rising above the block.
- **laser cutters** may never stay low when positioning, as only at free movement level DeskProto will switch off the laser.

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**. When for this value "true" is found, DeskProto will let the cutter rise to Free movement height for **every** positioning movement. Even when proceeding to the next toolpath. This hidden registry setting will overrule the automatic 'NeverStayLow' funtionality for the special cutters. When the string is "false" (or in fact any other text) nothing will happen.
This option cannot be checked in the user-interface, in order to use it you need to change this:
For Windows change registry key *HKCU\Software\Delft Spline Systems\DeskProto\8.0\Custom\NeverStayLow*
For MacOS change line "8·0.Custom.NeverStayLow" in file *DeskProto.plist*
For Linux change line "Custom\NeverStayLow" *in section "[8.0]" of file DeskProto.conf*.
Only edit the registry or these files when you are qualified and know exactly what you are doing !

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 a critical point in the process. Two separate options are offered. In both cases you can enter a percentage for the maximum feedrate reduction, 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) is removed. However, in certain cases the cutter has to remove material over its 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 moves much lower than during the previous toolpath, as shown in the small image next to this setting). The chip load is even higher in such cases as the chips cannot easily fly away 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).

When you reduce the feedrate, the **spindlespeed** may become too high to cut clean chips (instead the cutter will create dust and become very hot). This can be prevented by checking an option on tab Spindlespeed of the

Postprocessor: option 'Apply the Dynamic feedrate control percentages also on the Spindlespeed'. DeskProto then will reduce the spindlespeed too, with the same percentage.

The option Reduced feedrate for high chiploads is not available for all strategies.
Not for all cutters either: for hot wire cutting it cannot be applied, so it will then be disabled (grayed-out).
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 setting called *NumReducedInitialPaths*, which sets the number of initial toolpaths to be reduced. The setting will work only for strategies Parallel, Crosswise and Block.

It can be switched on by editing this setting and giving it a value from 1 (default) to 9:
For Windows change registry key *HKCU\Software\Delft Spline Systems\DeskProto\8.0\Custom\NumReducedInitialPaths*
For MacOS change line "8·0.Custom.NumReducedInitialPaths" in file *DeskProto.plist*
For Linux change line "Custom\NumReducedInitialPaths" *in section "[8.0]" of file DeskProto.conf*.
Only edit the registry or these files 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 you machine the Picture frame sample geometry using strategy 'Parallel to X', skipping the empty inside of the frame. The cutter then 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.

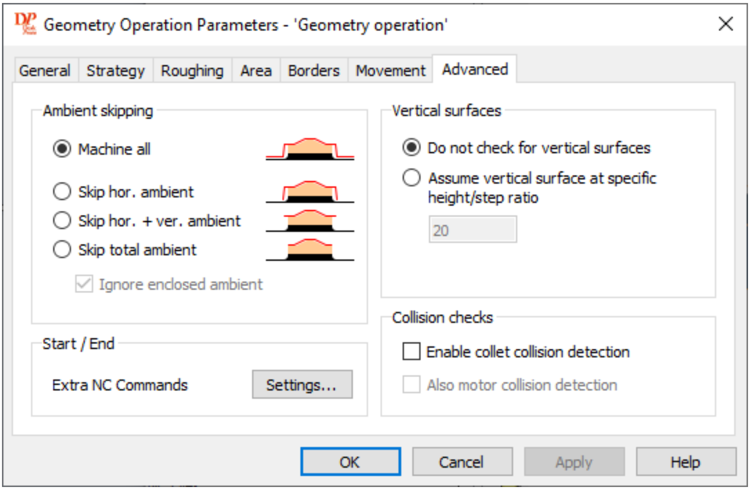
Background information about the sorting algorithm that is used, written by the programmer:
In DeskProto each toolpath is a series of small lines (linear interpolations, or G1 movements), where each movement is a straight line between two

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]: this is the area where no geometry is present. Basically DeskProto always machines a complete rectangular area, due to its parallel toolpaths approach. 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 edge of the area 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 images on this tab page illustrate the differences between these four skipping options.

For this skipping feature DeskProto will use the ambient area that is present on the minimum Z-level of the **operation area**. This may differ from the Ambient area as used in the Part parameters.

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.

Start / End offers the option to add extra commands to the NC program before the operation's 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, or to insert a comment line that mentions the name of the next cutter to be loaded. Button **Settings...** will open the Operation Start/End settings dialog.

Vertical surfaces

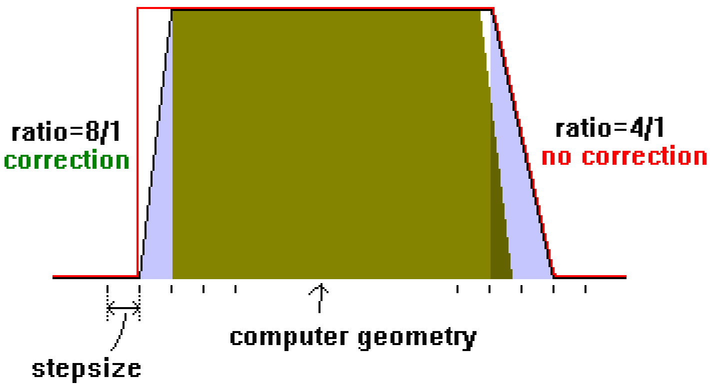
The algorithm used by DeskProto to calculate toolpaths does not really support true **Vertical surfaces**. DeskProto calculates its toolpaths based on a Z-Grid, connecting points on this grid to form the toolpath. For each XY position in the grid only one Z value is available, which means that vertical 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 will be machined having a small angle in the resulting part.
BTW this is not true for all strategies: a waterline toolpath does not have any vertical components. For waterlines this option still can be used though: when traveling to the next waterline a movement at an angle is present. The horizontal component of this movement is the Distance between toolpaths.

A simpler option to protect the vertical surfaces is present on the Roughing tab. When that option has been checked the option on the Advanced tab will be disabled, as it would be overruled by the Roughing setting.

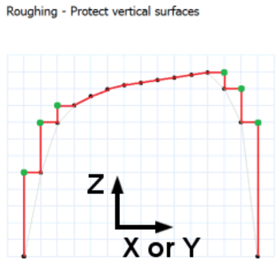
In case you need your model to have true vertical surfaces (and you do not want to use waterlines) you can achieve that by using and fine-tuning this option. You can let DeskProto assume a vertical surface in case the ‘toolpath-line’ is steeper than a certain angle. The angle is defined by the Ratio 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.

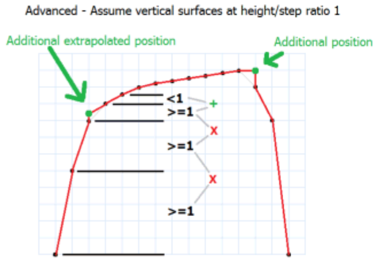
For every movement DeskProto will check this ratio, and for a movement that exceeds the ratio (so when the movement is steeper), DeskProto will insert an intermediate point: the movement will be split into a horizontal and a vertical component, to be executed sequentially. The result will be a vertical surface in the part, which otherwise would have been angled.

Example 1: **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 apply 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 at this point, 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.

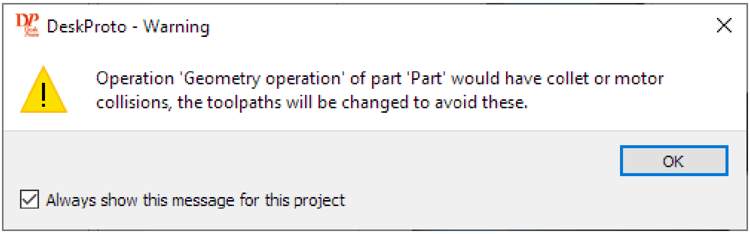
For parts without vertical surfaces this functionality is not needed, so for such models choose **Do not check for vertical surfaces**.

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. The toolpath over this vertical surface now contains a horizontal component of 1 mm (the stepsize) - however, you want it 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.

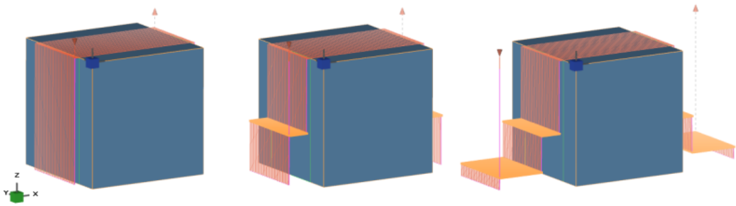
Collision checks



The two **Collision check** options are very useful for high parts, especially with high vertical or steep surfaces. With such parts a problem will occur when the vertical wall is higher than the free length of the cutter: the milling machine's collet will then damage the part (DeskProto only compensates for the geometry of the cutter, not for the machine-parts that hold the cutter).
Checking option **Enable collet collision detection** means that DeskProto will let the tool move away from the part in such case, preventing the collet to collide with the part. See the illustration below. 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.

DeskProto will let you know when collisions have been found and prevented, using the warning dialog shown above. This warning is optional (stored as setting for only this project), when it is no longer shown it can be restored in the Project parameters, on tab Advanced. The toolpaths that have been changed will be drawn orange.

The diameter of the collet can be defined at the Machine parameters dialog. .

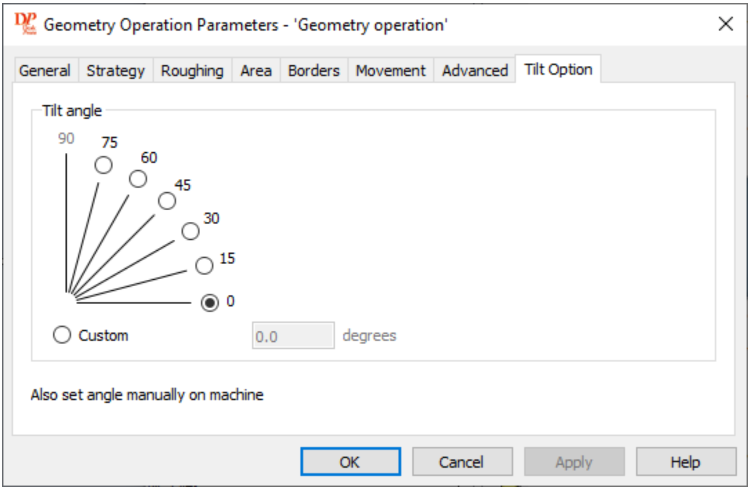


The option **Also motor collison detection** makes DeskProto perform the same trick for the bottom of the spindle motor. This checkbox is enabled only when the collet collision detection is enabled ánd when the motor dimensions have been set in the Machine definition.

The illustration above shows the results of the collision check options, for the most basic geometry that is possible: a cube of 50 x 50 x 50 mm. The cutter is a flat cutter, diameter 4 mm, free length 30 mm (so the cutter sticks out 30 mm below the collet). Left the toolpaths without collision detection (the collet will now severely damage the part). In the middle the toolpaths when the collet collision detection is enabled. Note the orange color that clearly indicates where the toolpaths have been changed to avoid collisions. The image on the right shows the result when the Motor collision detection has been enabled as well. This is for a motor diameter of 60 mm and a toolholder height of 15 mm

The orange toolpaths shown above will not always be completely present, as they may fall outside the material block and outside the Area to be machined. DeskProto normally adds a Border area ("Extra for cutter") around the Area, however this border only adds the cutter radius, so most orange toolpaths will fall outside.

Tilt option parameters

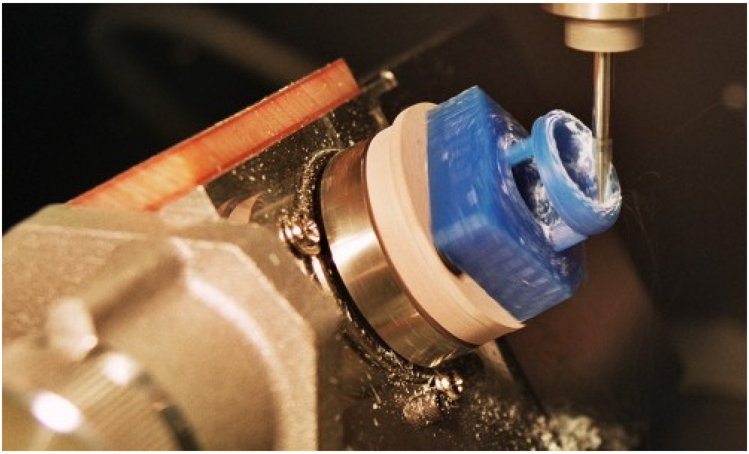


The rotation axis **Tilt option** is a feature where the complete rotation axis can be oriented in an upward angle, like the road surface of a drawbridge. While the axis is tilted it still can be controlled as 4th axis and process rotary toolpaths. This allows the cutter to also machine the inside of a ring: see the illustration below. Such tilting rotation is in fact a second rotation axis: a 5th axis (B-axis). So in the machine definition it can be found in the Rotary machine settings. Two versions are possible: a CNC controlled 5th axis, and a tilt option that has to be set manually.

Manual tilt option

The illustration below shows the **manually controlled tilt option** that is present on some wax milling machines for jewelers. Shown is the Roland JWX-10 "Jewela" machine.

The Tilt Option tab is visible only in case in the Part parameters the option "Use rotation axis tilt option" or "Use 5th axis as rotation axis tilt option" has been checked (and in case the machine has a 5th axis configured).



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.
Keep in mind that you need to manually set this machine to the same angle as entered in this dialog.

CNC-controlled 5th axis
For 5-axis CNC machines that have a **CNC-controlled 5th axis**, DeskProto supports only trunnion style machines, as explained in the Rotary machine settings. On such machine the large rotation axis can be used to orient the small rotation axis in a tilted position. So this tilt option can also be used on

these machines. Provided that you can indeed use the small rotation axis as 4th axis.
Important is that DeskProto will not automatically write the rotation command for the fifth axis (setting the tilt angle) to the NC file: you will have do add that by entering it in the operation's Start/End commands.

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 working with the same Workpiece zero point).

The Tilt option is available only for Geometry operations, so not for Vector and Bitmap.

4.2.3.3 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.

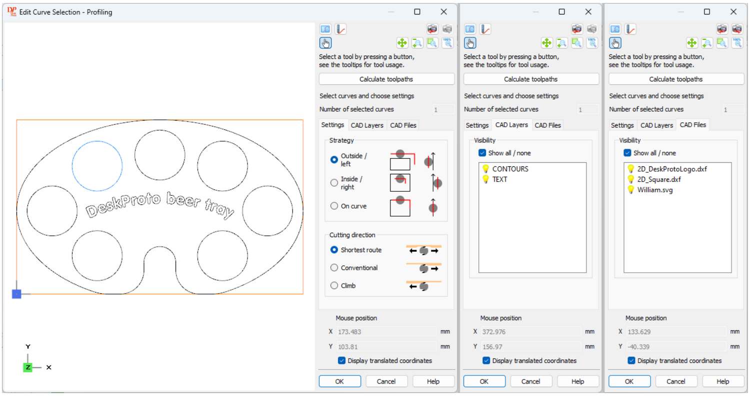
The Bitmap Operation parameters are identical to the Geometry Operation parameters (except for the Tilt option: that is not available for bitmap). The Bitmap Operation applies them on the Bitmap relief, the Geometry Operation applies them on the Geometry.

So **for information on any of the Bitmap tab pages, jump to these Geometry operation Help pages:**
General
Strategy
Roughing
Area
Borders
Movement
Advanced

This dialog can be reached via the Parameters menu, third option.
Or you can double-click on an bitmap operation line in the project tree (one of the third level items).
Or right-click on a bitmap operation line 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.

4.2.3.4 Select Curves

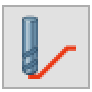


Purpose of this dialog is to select the vector curves to be used for this operation (or to edit that selection).
It is present in four different versions: once for **Profiling**, once for **Pocketing**, once for **V-Carving** and once for **Drilling**. All four versions offer the same functionality, 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, the V-Carving tab and the Drilling tab of the Vector Operation parameters.

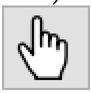
A top view drawing is present (no other views), in which all loaded curves will be drawn in **gray**.

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 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. These are of course active only when the view has been changed (Previous view) and when the Previous view button has ben used (Next View).



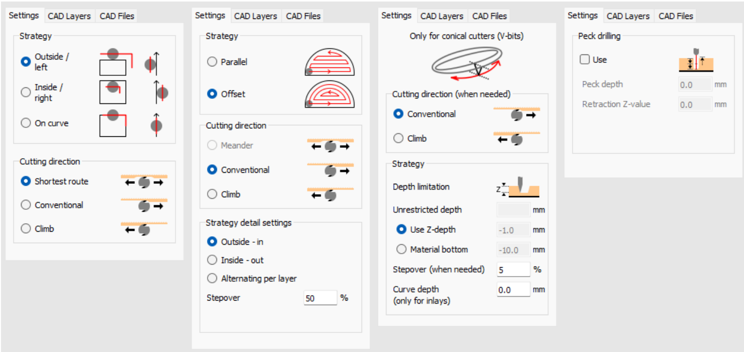
The leftmost button on the second row is the **Select** button. It is one of the five buttons to set which Mouse function has to be used in this dialog. Four of these are known from the Toolbar, this leftmost button is the **most important button in this dialog**. When it is active you can select a curve 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 **light blue**, for "selected".
Extra options when using the Select button:

- Draw a box from left to right **around** multiple curves to select them all
- Draw a box from right to left **over** multiple curves to select them all
- Keep the Shift key (on the keyboard) pressed while clicking to keep adding curves to the selection
- Keep the Ctrl key (on the keyboard) pressed while clicking to remove curves from the selection (on a Mac use the Command key instead of Ctrl)
- Type Ctrl-A to select all curves.

Button **Calculate toolpaths** (large button with text) will calculate and show toolpaths for the selected curves (just as the button with an icon).

Below this button DeskProto will show how many curves have been selected.

Not all curves can be selected:
- Profiling will not accept single points
- Pocketing and V-Carving will accept only closed curves
- Drilling will accept only single points, +-signs sized equal to the cutter diameter and circles that (almost) equal the cutter diameter.
Curves that are not acceptable will be drawn in **light gray**, for "can not be selected".
Vertical lines in the vector data (curves with all points having the same X and Y, and different Z values) cannot be selected in any of these four toolpath types.

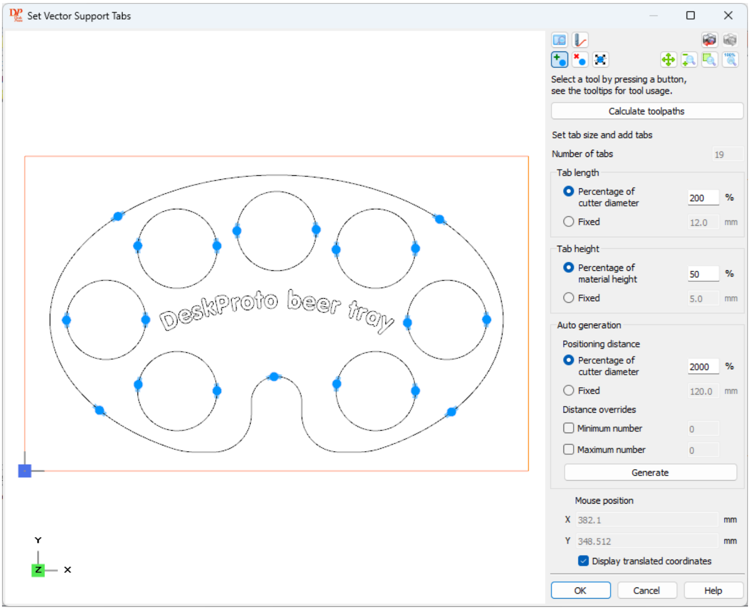


Under tab **Settings** you can set the most important parameters for that toolpath type: the above image from left to right shows the settings for Profiling, Pocketing, V-Carving and Drilling. These are the same as present in the Vector Operation parameters, making them also available in this dialog will allow you to quickly see the resulting toolpaths for any setting. For the Help information about these settings click on the above link for that toolpath type.

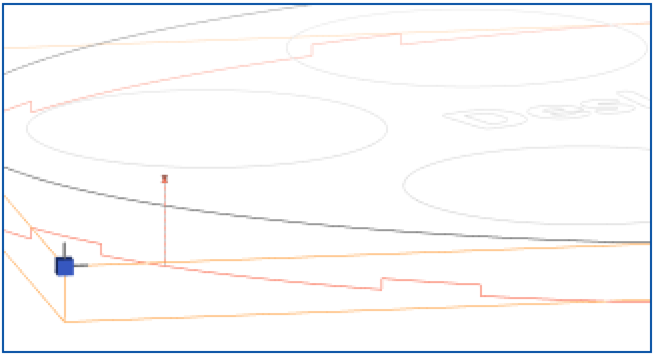
A very handy tool is available under tab **CAD Layers**. Here all layers in the available Vector data are listed (if any), with a light bulb icon that can be used so show or hide the curves in that layer.
Many DXF files contain layers, meant to make it easier to access the data by viewing at it layer 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 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. Remember that the light bulbs only set the visibility in this dialog, you still need to select curves with the mouse.
Note that these CAD layers are completely unrelated to the Roughing layers that DeskProto uses when roughing.

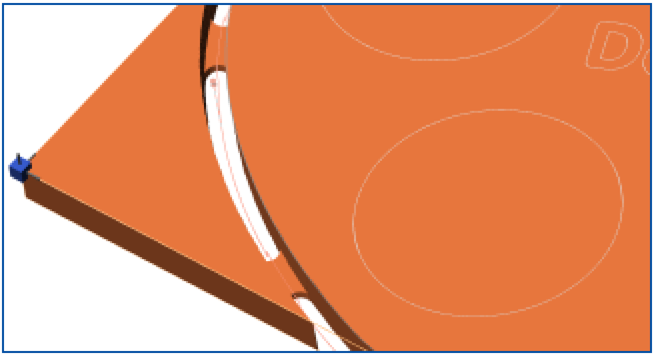
A similar list with light bulbs is shown under tab **CAD Files**. This list contains all files with vector data that you have loaded, and the light bulbs make it possible to select from which vector files the curves will be shown.

4.2.3.5 Set Vector Support 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 toolpath 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.





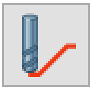
DeskProto offers **Support tabs** to solve this problem. Such tab keeps both sides of the toolpath connected by raising the cutter for a small section of the toolpath, leaving a small bridge of material that later can be removed manually. The two images above clearly illustrate how this works: left the toolpath, right a simulation.
In the first image these tabs are shown: the dark blue dot shows the center of the tab, the light blue area indicates the length of the tab.

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. Vector support tabs are available only for vector toolpaths without Z-values (2D curves). So they cannot be used when a 2D curves is projected on a 3D geometry, or when for a 2D vector file the option "Use Z-values" has been checked in the project parameters.

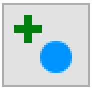
The dialog **Set Vector Support Tabs** allows you to set the number of tabs, the location of each tab and the tab size. You can open this dialog on the Profiling page of the Vector Operation Parameters. The drawing will show a top view of all vector curves, with the selected curves drawn in black. Support 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:

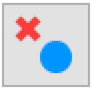
Button to open dialog Items visible for Set Graphically, in which you can select which items should and should not be included in this drawing. It will be clear that item "Vector curves" now cannot be switched off.



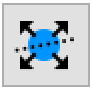
Button Show Toolpaths will show (and if needed first calculate) the toolpaths in this top-view drawing. Pressing the button again will again remove the toolpaths from the drawing.



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 dots, 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 dots, 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. In the image the length of each tab is shown by a light blue background color round the curve.
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 parameters in the **Auto generation** group:

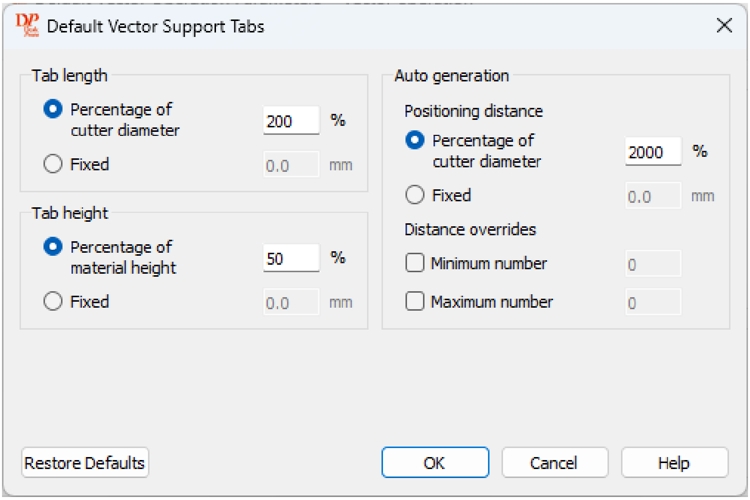
Positioning distance
The distance between two tabs: DeskProto will automatically set tabs along each selected profile using this distance. Here again you can either use a **percentage** (of the cuter diameter) or a **fixed** value.

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
This button will make DeskProto remove all current tabs and set a complete series of tabs based on the parameters that you selected.

4.2.3.6 Default Vector Support Tabs

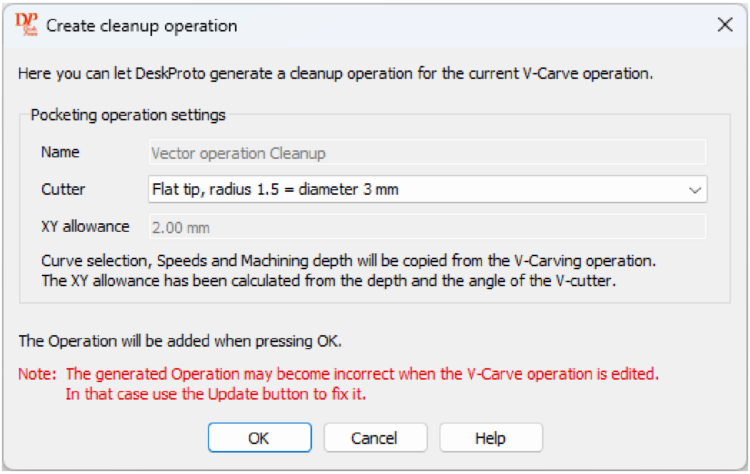


The use of Vector support tabs is explained on the help page for dialog Set Vector Support tabs.
In that dialog they can be set manually or they can be generated automatically using the auto positioning parameters.

On the Profiling page of the Vector operation parameters it is also possible to select **Default tabs**. This dialog Default Vector Support tabs allows you to set the parameters to be used when generating these default support tabs.
It can be accessed via the Default Vector Operation parameters

The parameters are the same as in dialog Set Vector Support tabs and you can find all relevant information on that Help page.

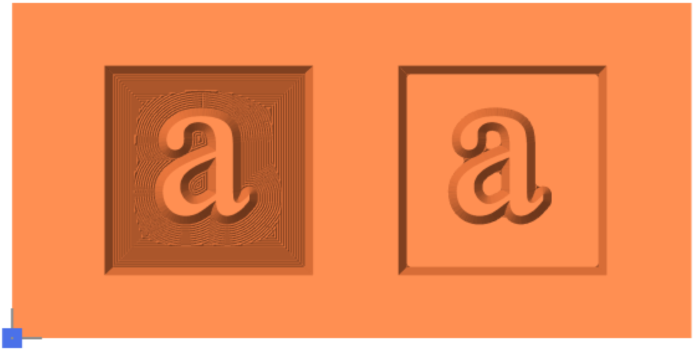
4.2.3.7 Create Cleanup operation



A Cleanup operation is meant to clean the flat bottom of a V-Carved groove. In a V-Carving operation such groove is machined by a V-Cutter with a sharp tip, so this bottom will not be a flat surface: it is completely filled with the cusps (small ridges) between the V-Cutter toolpaths. See the illustration below.
This dialog is meant to create an extra operation that will cleanup this bottom: make it a flat surface using a flat cutter.

You can reach this dialog using button "Create cleanup operation..." on the V-Carving tab of the Vector operation parameters.
Once a cleanup operation has been created for this V-Carving operation the name of the button will change to "Update cleanup operation". After pressing the Update button this dialog won't be shown again: only the settings of the cleanup operation will be updated to match any changes in the V-carving operation.

The cleanup operation of course only is effective when a flat bottom is present in the V-grooves. And the flat area also must be so large that the flat cutter that you select can indeed be used to make it flat. If not DeskProto will show an error when you calculate toolpaths: "*The operation 'Vector operation Cleanup' of part 'Part' has no valid toolpaths*".

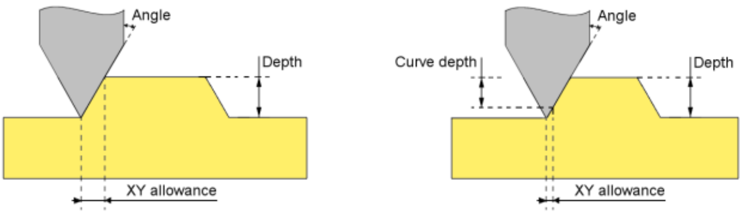


On the left side of this simulation you see the result of the V-Carving toolpaths for this design (Times New Roman font, 45 degree V-bit, depth limitation -2 mm). On the right side the result after also using the cleanup operation (flat cutter D 2 mm). Note that this flat cutter cannot completely cleanup the bottom surface: it cannot machine the sharp inner corners.

The new operation will be a Pocketing operation, for which three settings are shown in this dialog:

The **Name** of the new operation.
This is the name of the current V-Carving operation with the word "Cleanup" added. This naming convention is how DeskProto knows that these two operations are connected, so do not rename (or rename both).

The **Cutter** to be used in the new operation.
Here your can only select one of your cutters with a flat top: using a different cutter type would not create a clean flat bottom.



The **XY allowance** (that will be set on the Roughing tab of the new operation) is the distance that the flat cutter needs to keep away from the curves. It is calculated from the **Angle** of the V-cutter and the **Depth** of the V-carved groove (note that in DeskProto this angle is the grinding angle, not the included angle). The formula is simple:
`XY allowance = Depth * tan(Angle).`

The above image on the right shows the situation when a curve depth has been set (for inlays). Now the formula is:
`XY allowance = (Depth - Curve depth) * tan(Angle).`

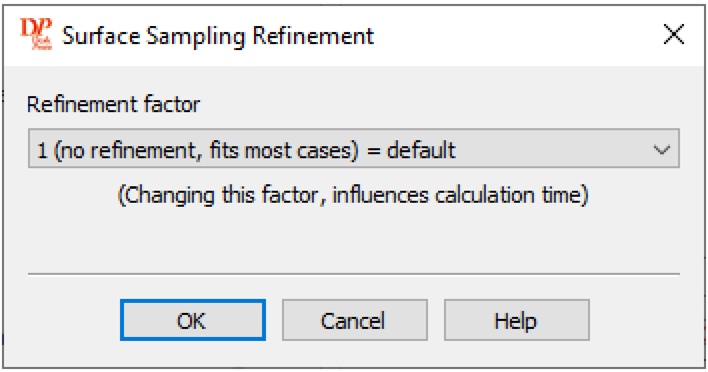
No need to worry about these formulas as DeskProto takes care of these calculations.

4.2.3.8 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 the highest Z-value of all the geometry within that cell will be stored: the surface is sampled. In other words: the DeskProto algorithm is based on **Surface Sampling**.

In the Geometry Operation parameters you enter the Precision by setting the “Distance between toolpaths” and the “Stepsize along toolpath”. The smallest of the two will be used as Grid-size (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 Grid-size. This is called subsampling or sampling refinement, as for each toolpath point more than one grid-cell will be used. 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).

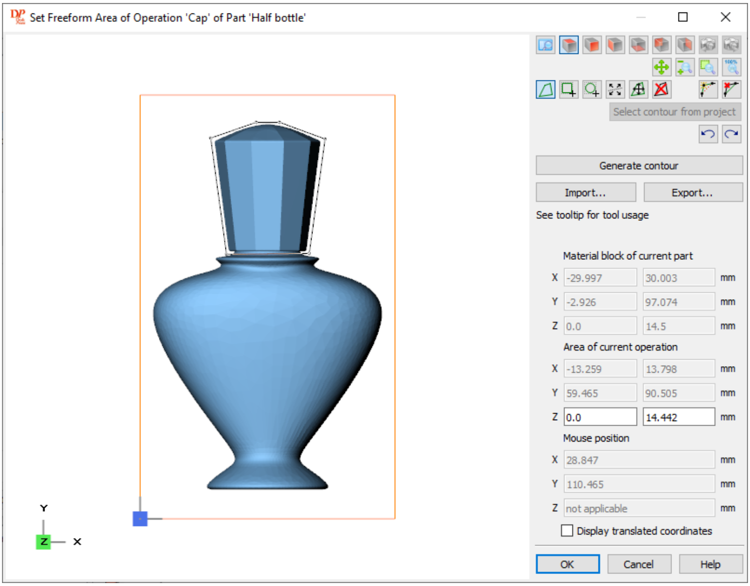


The Surface Sampling Refinement dialog allows you to fine-tune the grid-size or calculation precision. The higher the Refinement Factor, the more accurate the toolpaths and the longer the calculation time (quadratic). A higher factor also means that DeskProto will need more internal memory (RAM). A refinement factor 1 will result in a Z-grid with a cell-size that equals the settings for Distance between toolpaths and Stepsize along toolpath. A **Factor 2** will halve the cell-size and thus 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.

4.2.3.9 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 resulting volume 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.

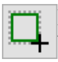
On the right side of the drawing four rows of buttons are present.

 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 (aligned right) 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). You can see which button is active as it has a light blue background (in the image the leftmost button on the third row).

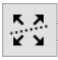
The eight buttons on the third row are special for this dialog: these are the drawing tools used to graphically draw and/or change the area. These are *mouse function buttons* as well: of the mouse function buttons (13 in total) only one is active at any time. You will see that the shape of the cursor changes when you select a different mouse function button. Each button comes with a tooltip text that explains what this tool can do.

 **Draw a Polygon** as new Freeform contour: each mouse click will add one point. The polyline that you draw is always closed, therefore it is calle Polygon. End the function with a right mouse-click. Extra key functions:
ALT (on Apple the Option key): snap points to geometry points
SHIFT: create lines at 15° angles
CTRL (on Apple the Command key): create lines at 1° angles
CTRL + SHIFT: create lines at 5° angles

 **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.
SHIFT: When you press the shift button during this input DeskProto will lock horizontal and vertical 1:1, forcing a square.

 **Draw an Ellipse/Circle** as new Freeform contour: click the left mouse button, move the mouse and release. The ellipse or circle that is drawn is in fact a polygon.
SHIFT: When you press the shift button during this input DeskProto will lock horizontal and vertical 1:1, forcing a circle.
When you activate this tool by pressing the button you will see that an extra input field appears:

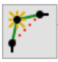
the **Number of points** that you enter here is the number of points that will be used for this polygon. For low numbers the resulting boundary will not be a circle.



Adjust Boundary of a Freeform contour: you can move the points and the sides of the contour by dragging with the mouse. You can also resize and rotate the complete polygon. Watch the cursor to see if you will drag a point, drag a line or (when clicking outside the polygon) rotate the polygon.
Extra key functions for points:
ALT (on Apple the Option key): snap points to geometry points
CTRL (on Apple the Command key): resize the polygon
SHIFT: resize the polygon and keep the aspect ratio
SHIFT+CTRL: resize the polygon and keep, keep X or Y size.
Extra key functions when clicking outside points:
SHIFT: create lines at 15° angles
CTRL (on Apple the Command key): create lines at 1° angles
CTRL + SHIFT: create lines at 5° angles

Move a Freeform contour: simply pick and drag the complete contour line.

Delete a Freeform contour.



Add new point to the Freeform contour: click with your mouse to add a point. The side of the polygon will be split up into two new sides.
ALT (on Apple the Option key): snap points to geometry points

Delete point from the 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.

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 (button Adjust Boundary). 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 activated 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 large button "Select contour from project" is the thirteenth *mouse function button*, so when selected it will have a blue background.

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

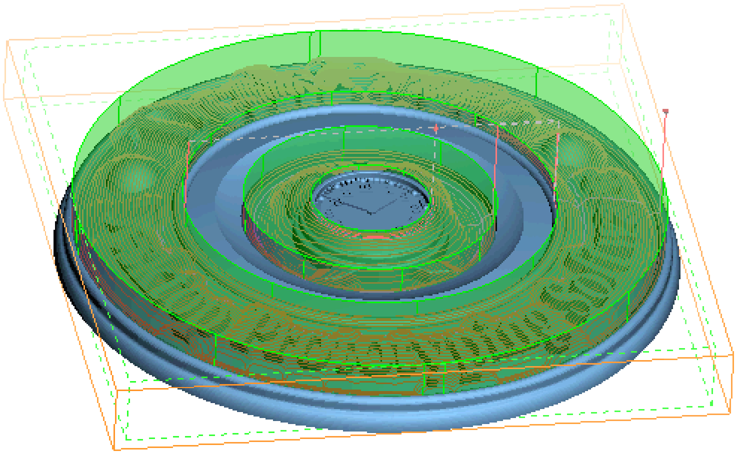
Nine levels of Undo are supported plus one level of Redo, so a total of 10 situations is stored.

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.

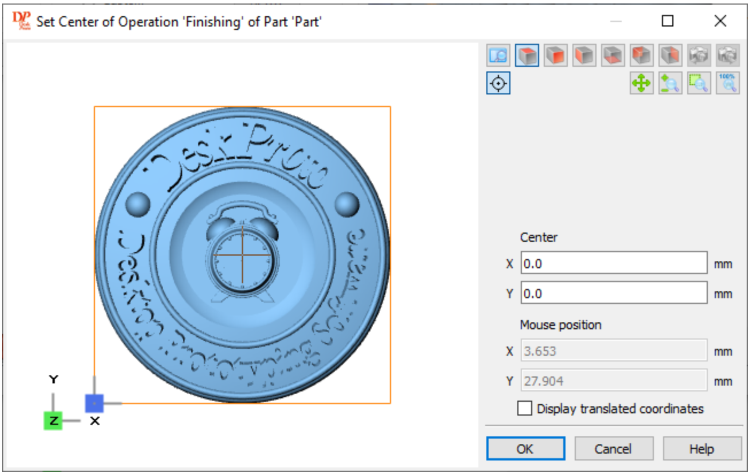
A very powerful option is offered by button **Generate contour**: DeskProto then will automatically generate a freeform area contour that follows the outside contour of the current geometry. This will also work for multiple geometries, and for geometries that contain holes. It will also work for a bitmap relief, though then first the Bitmap Contour tolerance dialog will pop up, as for bitmap data some extra settings are needed.

This contour line to define the freeform area can be imported from a 2D vector file. The button **Import…** 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.
The reverse action **Export...** will save the current freeform contour(s) in a vector file.

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.



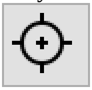
4.2.3.10 Set Center



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 Detail settings, where you can enter the X and Y coordinates for this center point.

At that same location (Operation parameters, tab Strategy) 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.

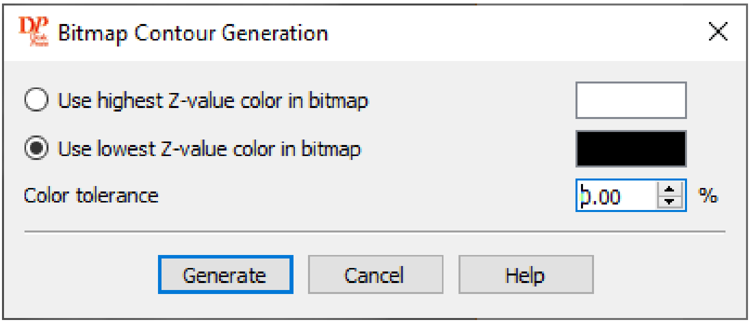
The dialog is in fact almost identical to the other dialogs for graphical input: Set Area and Set Freeform Area.
Only the functionality is different, and is in fact very limited:



The 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.

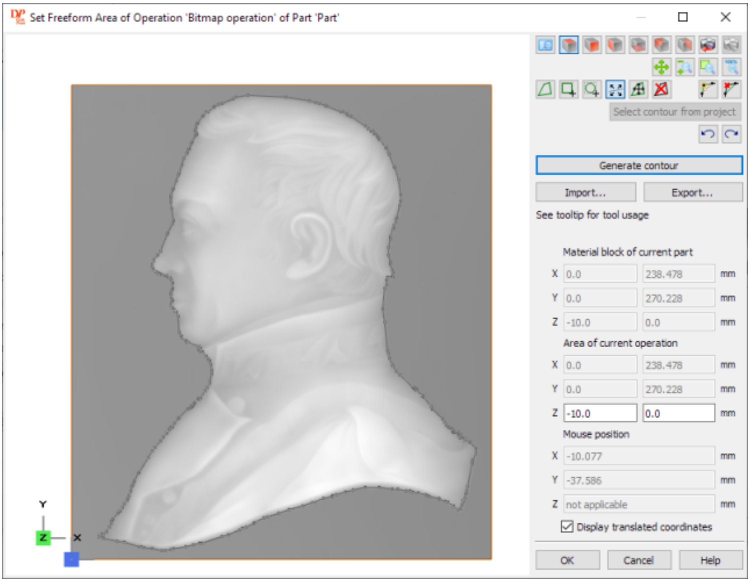
4.2.3.11 Bitmap Contour Tolerance



The dialog to set a Freeform area offers the option to **generate a contour**. For a geometry operation that is a contour line around the complete geometry. However, for a bitmap operation a contour round the complete bitmap is not useful as that will be a simple rectangle. So for a bitmap operation DeskProto offers a more refined tool, in which you can set the contour round all pixels below or above a certain gray level. The idea is to create a contour round the actual relief, excluding the background (the bitmap's 'ambient'). In this dialog the threshold gray level can be set.

For a negative relief the backgrond (the "ambient area") will be the color with the highest Z-value, for a positive relief it will be the color with the lowest Z-value. Which of these two is black and which is white depends on the Part parameters Z-settings that you made for this bitmap.

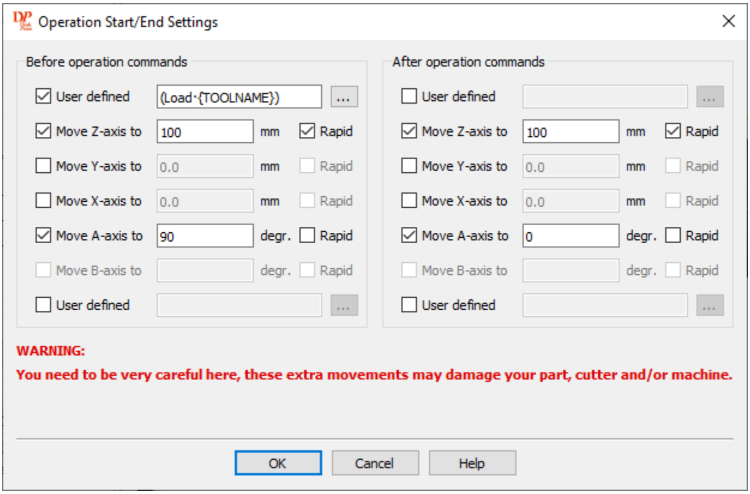
However, the background of the image in many cases is not just pure black or pure white: it will also include almost black pixels, and almost white respectively. That can be solved by entering a value for the **Color tolerance**. The value that you enter as Color tolerance is a percentage. Of all gray values between black and white, which percentage do you want included in the freeform area. 0% means that only pure black or pure white will be considered as background when generating the contour. In practice you will have to experiment a bit to find the tolerance that produces the contour that you need.



An example: the sample bitmap file Radetzky.jpg that is included in every DeskProto installation.
The background color of this image is not pure black, so selecting black and a Color tolerance of 0% or 1% does not result in a contour line around the relief.

In order to get the contour line shown in the image it was needed to experiment with the tolerance value: this contour was generated by selecting white and a Color tolerance of 89%. BTW exactly the same result is achieved by selecting black and a color tolerance of 11%.

4.2.3.12 Operation Start/End Settings



This dialog is meant for advanced users only, as you can conclude from the warning in red.
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) and at the End (after its toolpath). For instance to let the cutter travel to some safe position after finishing the operation. Or to let a rotation 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** (Start commands) can be specified. These will be written as extra lines in the NC program file, just before the toolpath of this operation. 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*. A powerful option in a user defined command is the use of **placeholders**, see below.
- **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 Y, the X, the A, and finally the B 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.

As the edit field for user defined commands is very small an extra button **[...]** is present on that line. This will open the NC code dialog, that offers much more room to add and edit command lines, and also an option to show the result of the placeholders ('Show example'). As said, e advise that are used only by advanced users !
For each of the Move commands an extra checkbox **Rapid** is present: checking that box means that the movement will be done in rapid mode.

The screenshot above shows a few possible uses:

- Two rotation commands are used for **indexed machining**: machining a part from several sides: three-axis toolpaths 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 'parts', each time with a 90 degree rotation in-between. It also applies a Z-axis command, to make the cutter move upward to a safe height before the A-axis starts rotating, as otherwise the block might hit the cutter while rotating. The toolpaths for all operations (in several parts) then can be written to one large NC file using Chaining.

- In the four edit boxes for User defined commands you can use **placeholders**. These are special bits of text that you can enter, which will be replaced with some parameter setting when the NC program file is written. For instance the text "`{TOOLNAME}`" will be replaced by the name of the cutter that is needed for this operation.
A placeholder that forces a new line in the NC program is available as well. A list of all available placeholders can be found at page Postprocessor placeholders (despite the name they can also be used in this dialog).
For machines that support the use of **comment lines** in the NC code the User defined fields is can of course also be used to enter comment lines.
When your machine supports **macros** you can use the User defined field to call a macro. We have for instance seen a macro to reset the A-axis to a value between 0 and 360 after completing a helix toolpath.

The screenshot shows adding a comment line (this machine considers any text between bracket as comment) with the name of the cutter to be loaded. When the CNC-controller shows this comment on screen the operator will know which cutter needs to be used.

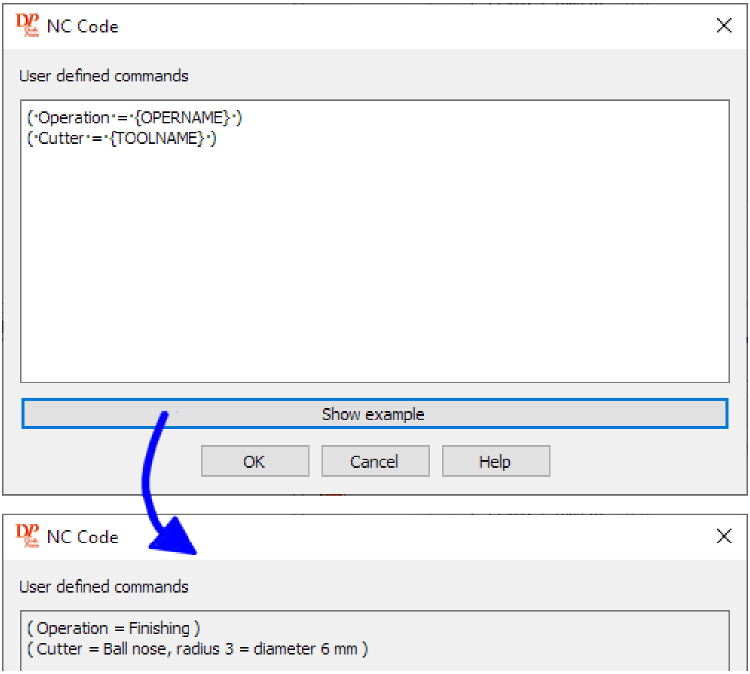
In the four "User defined" edit fields a **space** will be shown as a "**middle dot**" (·), like in the example screenshot after "Load". This is done as otherwise spaces would easily be overlooked. When editing this field you will again see a space.

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.

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. DeskProto cannot simply use the current position for the other two axes, as at the start of the toolpath it does no know the current position. 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 they will be grayed out. So the illustration above is made with a four-axis machine selected: the B-axis is not available.

4.2.3.13 NC code



This dialog is a sub-dialog of the Operation Start/End settings dialog. You can reach it by pressing the small button **[...]** that is shown next to each User Defined edit field. Its purpose is to provide a larger area for editing these User defined commands.

User defined commands can be used to enter **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* (except the placeholders, see below).
We advise that user defined commands are inserted only by advanced users !!

The image above shows the use of two "**Placeholders**": {OPERNAME} and {TOOLNAME}. These are special bits of text that you can enter, which will be replaced with a parameter value when the NC program file is written. For instance the text "`{TOOLNAME}`" will be replaced by the name of the cutter that is needed for this operation.

A list of all available placeholders can be found at page Postprocessor placeholders (despite the name they can also be used in this dialog).
A placeholder that forces a new line in the NC program is available as well: {N}. DeskProto will automatically create such NewLine placeholder when you start a new line in this edit box.

When you press button **Show example** the dialog will show an example of the resulting NC code, as shown by the large arrow in the illustration above.

For machines that support the use of **comment lines** in the NC code the User defined fields can be used to enter comment lines. The image above is for a machine that interprets lines between brackets as comment: these comment lines will tell the operator which operation will follow, and which cutter needs to be used. Other machines treat for instance all text after a semicolon (";") as comment.

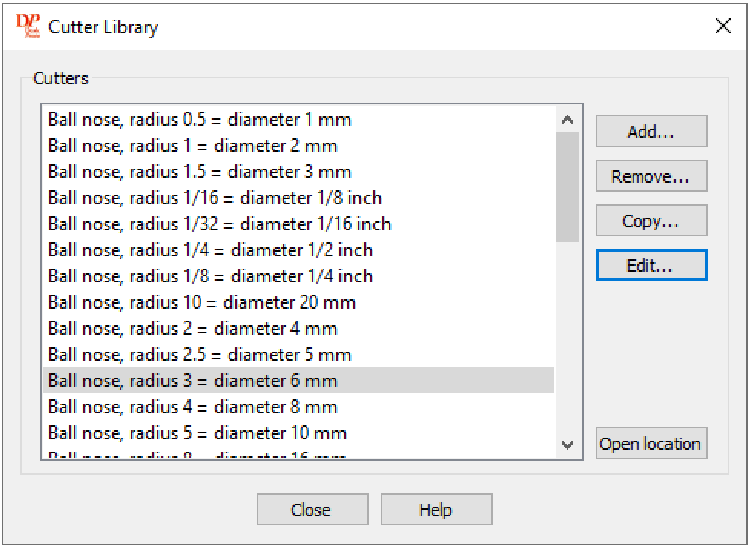
When your machine supports **macros** you can use the User defined field to call a macro. We have for instance seen a macro to reset the A-axis to a value between 0 and 360 after completing a helix toolpath.

When editing these User defined commands **spaces** will be shown as a "**middle dots**" (·), like in the example screenshot on both sides of the "=". This is done as otherwise spaces would easily be overlooked. When pressing Show example the actual spaces will be shown.

4.3 Libraries

4.3.1 Library

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:



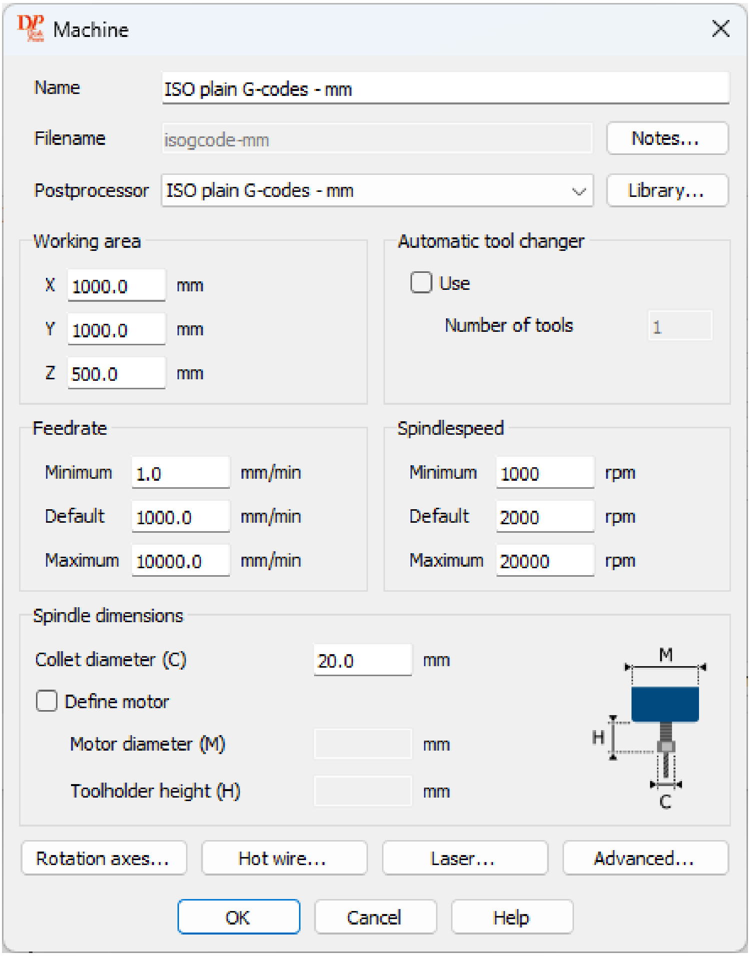
A library is just a collection of definitions. You can either **Add**, **Remove**, **Copy** or **Edit** an item. The definitions are saved to disk in the Drivers folder, which can be set at the General tab page of the Preferences dialog box. They are saved to disk when you click the **Close** button of this 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 your operating system to do so (for instance using Windows Explorer). 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.

Further 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. Button **Open location** will open that folder in Explorer/Finder/FileManager. All these files are plain ASCII text files, that can also be edited using a plain text editor.

4.3.2 Machine



All settings in the screenshot above are for a "generic" machine definition, to be used for any machine that runs on ISO G-codes. When you use this machine definition you will need to edit the speeds and dimensions 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 add, remove and copy machines by adding/removing/copying MCH files to/from the DeskProto Drivers folder (as set in the Preferences). The Drivers folder can be found by pressing cancel here and then button 'Open location' in the underlying dialog.

Button **Notes...** will open the Notes dialog, in which you can enter extra information that you want to store in this machine definition. Details like when it was created, and who created it.

The **Postprocessor** that 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. Button **Library...** will open this postprocessor library.

Most other values that you enter in the Machine dialog are in fact less important, as they will be used only to check whether the parameters that you will enter for a project do not exceed the machines' capabilities. So most of these values do not influence the resulting toolpath (only the Spindle dimensions do, and some of the Rotary/Laser/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. Note that these dimensions often do not match the size of the working table: for many machine the table is larger than the working area.

Check the option **Use** for **Automatic Tool Changer** (ATC) when your machine has such ATC. The **Number of tool**s is the number of tools that this machine can store and select automatically.
Normally this applies only to machines that have an Automatic Tool Changer. However, some machine without an ATC still are able to process a toolchange command: by asking the operator to perform a manual toolchange. For such machines we advise to check this option and to set the Number of tools to 99. This number of tools value is used for validation only:

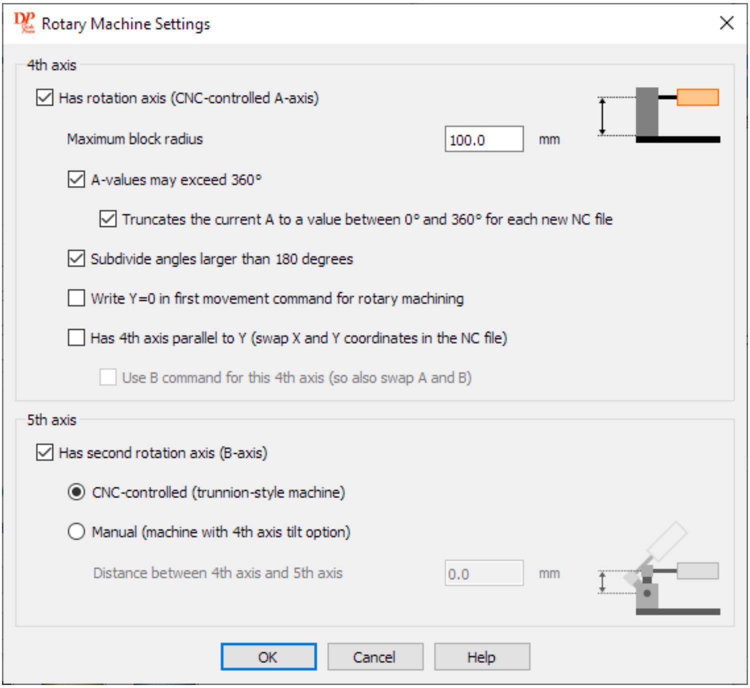
DeskProto will give an error when the cutter to be loaded has a number that is higher than this Number of tools.

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 for every new operation. And also when you select 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 the feedrate command for rapid movements, that rapid feedrate is determined by the maximum feedrate that is set here.
The units for the Feedrate can be set in the Preferences.

The Spindle **Dimensions** are used for the Collision checks that you can enable in the Operation parameters: toolpaths will be changed (in case needed) to prevent such collisions. The result is that some un-machined material will remain at these locations.
The **Collet diameter** is used for the collet collision detection: to prevent the collet from damaging the model.
The **Motor diameter** is used for the motor collision detection: this is a similar check for the spindle motor.
When you check **Define motor** you can enter dimensions for the **Motor diameter** and for the **Toolholder height**. The illustration in the dialog shows which dimensions need to be entered.

The **Rotation axes...** button leads to the Rotary machine settings.
The **Hot wire...** button leads to the Hot wire machine settings.
The **Laser...** button leads to the Laser machine settings.
The **Advanced...** button leads to the Advanced machine settings.

4.3.3 Rotary Machine Settings



This dialog is a part of the Machine definition of DeskProto, and can be reached via the Rotary Settings button on the Machine definition dialog. The rotary settings configure the availability and dimensions of the optional A and B rotation axes.

4th axis:
The fourth 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. When the rotation axis is parallel to X (as in DeskProto) it is called an A-axis.

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 **Maximum block radius** (in previous versions called 'Distance to working table of machine') is used to check if the block of material is not too large for the machine. For most desktop machines this 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 the radius) for which the block still can be rotated on this machine.

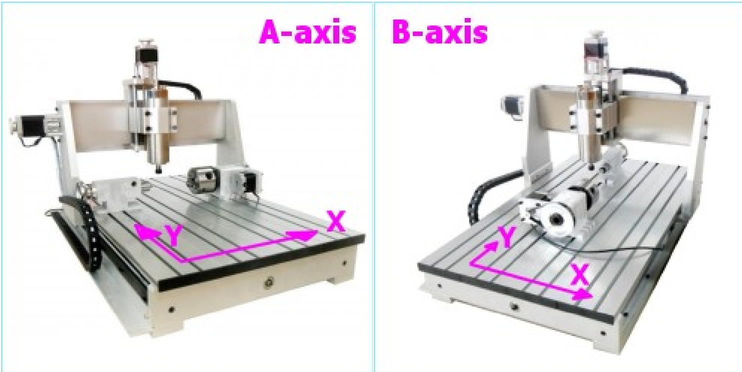
On some machines the rotation axis can only revolve a limited number of times, and has to then rewind. Either because of mechanical limitations or because of software limitations. Other machines allow you to keep rotating in one direction: for such machine you can check the option **A-values may exceed 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 (in degrees).
When this option is **not** checked the values for A (when setting the area to be machined) may not be lower than -360 degrees and not higher than 360 degrees. For such machines a helix toolpath is not possible.

As said above: when rotating in one direction all the time the final A-value can be a very high number. Normally DeskProto remembers that current position, so the next NC file (for instance for the next operation) will start with this very high value as A-position.
However, when the first NC file has been finished, some machines will automatically reset the A-position to the value between 0 and 360 that represents the same position. For instance A=770 then will be reset to A=50, as 770 = 2x 360 + 50. Other machines will keep the final A-value as current position.
When you your machine is a resetter you need to check the option to **Truncate to a value between 0 and 360 degrees for each new NC file**. Otherwise when you start the second NC file your machine will keep rotating for a long time in order to first reach that high A-value; as only then the actual toolpath can be started.

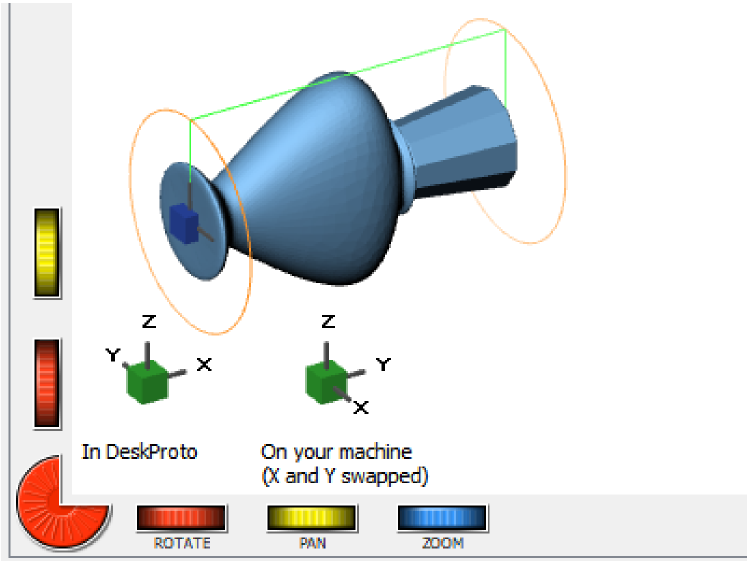
One more detail for which CNC machines with a 4th axis show different behavior is whether or not the machine always takes the shortest route. Say you have a toolpath where the subsequent steps are for points with these A-values: 0 | 30 | 60 | 270 | 300 | 330 | 360. Some machines will run this by keeping the same rotation direction (with increasing A-value) all the time, also when rotating from 60 to 270. Other machines however will say that decreasing A from 60 to -90 is a shorter route to exactly the same position, and will automatically select that route. Which may destroy your part in case that was not the intended toolpath. For such machines you need to check the option **Subdivide angles larger than 180 degrees**. DeskProto then in the example above simple inserts one extra point in the toolpath (halfway 60 and 270), making the intended route the shortest for each step in the toolpath.

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 (DeskProto cannot first insert a Z-movement as it does not know what Z-value to use). 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 default situation in DeskProto, which for a 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. All settings in DeskProto then are conform the left orientator, the NC file is conform the right orientator. These orientators are visible only when the item 'View orientator' has been checked in the Items visible dialog.

Note that when you would simply swap X and Y values the resulting coordinate system would change from right-handed to left-handed, and the machined parts would be mirror images of the parts in CAD. In order to prevent that DeskProto will reverse the sign of all new X-coordinates (multiply the original Y-value with -1.0) when this option is checked. For rotary toolpaths that won't make a difference as that axis is not used. For standard three-axis machining you need to correctly position the origin on your machine.
Only the coordinate values are swapped: the new X-coordinate value will be formatted using the settings in column X on tab movement of the postprocessor dialog, same for Y.

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.

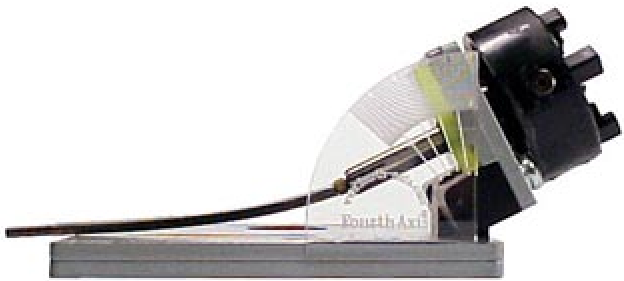
The **first** type of 5th axis is a CNC controlled B-axis: so an axis identical to the 4th axis, but then parallel to Y instead of parallel to X. You can enable such 5th axis in DeskProto by checking the option **Has second rotation axis (CNC-controlled B-axis)**. When both the 4th axis and this 5th axis are configured the result is a five-axis milling machine. DeskProto only supports five-axis machines where the part rotates. This is called a trunnion style 5-axis machine configuration, or a double swivel table configuration. Machines where the cutter rotates (swivel head) are not supported.

The photo above shows such **trunnion style** machine: this means that the two rotation axis units are built on top of one another. On the photo it seems as if the small rotation axis is parallel to Z. That can be easily fixed though: when the large rotation axis turns 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 where the two rotation axes intersect.*

In DeskProto this 5th axis can be used for indexed machining, using Start commands in the Operation parameters to issue the rotation commands for A and B. In the Tutorial manual and the instruction videos you can see how this works.

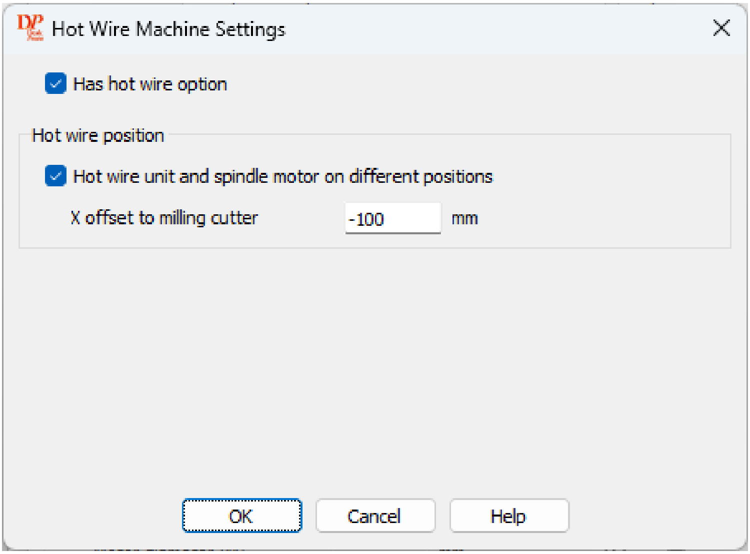
The **second** type of 5th axis is a special feature: the **Rotation axis tilt option**. Only a few machines support this (for instance the Roland JWX-10 and MDX-40). As the image on the dialog shows: the complete 4th axis unit will be tilted, like a draw-bridge being opened. As this rotation is around 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. On these two machines this tilt-angle needs to be manually set: unlock the rotation option, rotate to the desired angle and again lock it. The Tilt option tab of the Operation parameters includes a photo of the Roland JWX-10 machine carving a wax ring.

The tilt option can also be used on Trunnion-style 5-axis machines (as shown in the photo above). As said, when on this machine the large rotation axis rotates 90 degrees the small axis can be used as a standard rotation axis (the "fourth axis") in DeskProto. And with the large axis on say 45 degrees it can be used as tilted rotation axis. The tilt angle then needs to be set in the operation's Start/End commands to have it included in the NC file.



One extra parameter is available for the Tilt option: the **Distance to rotation axis,** which sets a vertical distance between the A-axis and the B-axis.This setting only needs to be used for machines where the A-axis and the B-axis do not intersect. Like in the illustration above (this is the Inside Ring Engrave Option by Fourth Axis® in Australia). For most machines (the two just mentioned and also the trunnion-style CNC-controlled machines) this distance is 0.0 and this setting can be ignored.

4.3.4 Hot Wire Machine Settings



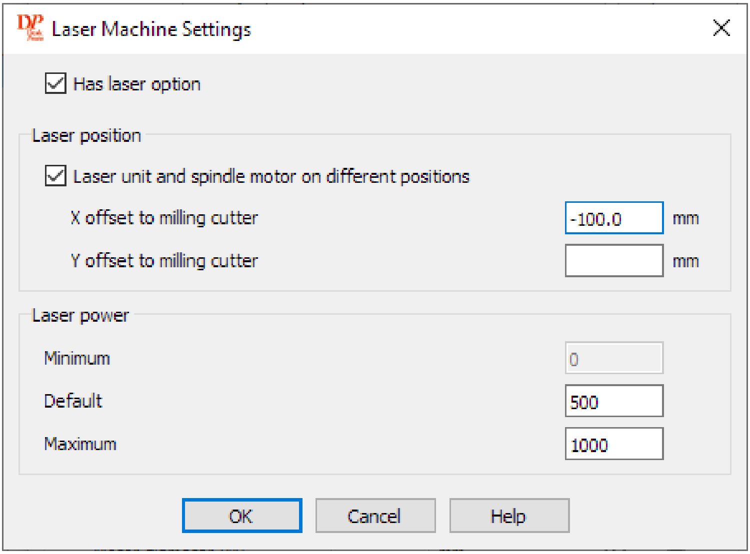
This dialog is a part of the Machine definition of DeskProto, and can be reached via the Hot wire Settings button on the Machine definition dialog. The hot wire settings configure the availability and characteristics of a **Hot wire cutting module** on your machine.

The option **Has hot wire option** needs to be checked in order to make hot wire cutting available for this machine. If not you will not be able to select a Hot wire cutter (in the Geometry operation parameters).
In order to use this option you need to make sure that hot wire settings have been configured in the postprocessor that is selected for this machine.

For machines that have the **Hot wire unit and spindle motor on different positions** (mostly machines where both options are permanently present) you can define the **Hot wire position** by entering an **X-offset** to the milling cutter. DeskProto calculates toolpath positions for the cutter, these offsets tell the program what to add or subtract for X to get the hot wire at that position.
In the above example the hot wire unit is located 100 mm to the left of the spindle motor's centerline (with an X-axis from left to right). That means that in order to get the hot wire on the same location as the cutter all X-coordinates in the NC file for a hot wire operation will be be corrected by adding 100 mm.

For machines without a spindle motor this option can remain unchecked.

4.3.5 Laser Machine Settings



This dialog is a part of the Machine definition of DeskProto, and can be reached via the Laser Settings button on the Machine definition dialog. The laser settings configure the availability and characteristics of a **Laser engraving module** on your machine.

The option **Has laser option** needs to be checked in order to make laser engraving available for this machine. If not you will not be able to select a Laser cutter (in the Vector operation parameters).
In order to use this option you need to make sure that laser settings have been configured in the postprocessor that is selected for this machine.

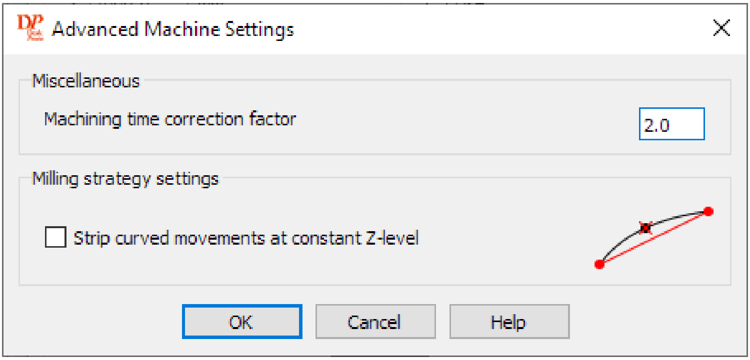
For machines that have the **Laser unit and spindle motor on different positions** (mostly machines where both options are permanently present) you can define the **Laser position** by entering an **X-offset** and a **Y-offset** to the milling cutter. DeskProto calculates toolpath positions for the cutter, these offsets tell the program what to add or subtract for X and Y to get the laser unit at that position.
In the above example the laser unit is located 100 mm to the left of the spindle motor's centerline (with an X-axis from left to right). That means that in order to get the laser beam on the same location as the cutter all X-

coordinates in the NC file for a laser operation will be be corrected by adding 100 mm.

For machines where the laser unit replaces the spindle motor this option can remain unchecked.

The values that you need to enter for **Laser power** in most case are NOT the actual power (in Watts). Most laser controllers use a fixed range (independent of the actual power), like from 0 to 255 or from 0 to 1000. Other controllers prefer a percentage, so values from 0 to 100. Consult the documentation of your laser unit to find out what it needs.
The minimum power value of 0 cannot be changed, as that will be used to switch the laser off.
The values that you enter here are used when setting the S-value in the Vector Operation parameters.

4.3.6 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 **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, as the actual machine time cannot be smaller that the theoretical time. You can find the explication why such conversion may be needed on page Estimated Machining time.

The second 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 DeskProto 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 be calculated as a series of such small steps, though for such straight movement in fact all intermediate points will 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.

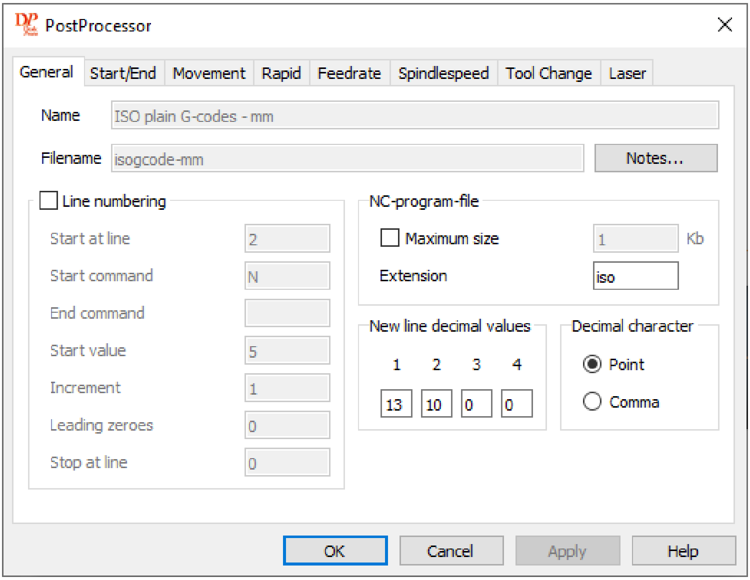
For some other machines checking this option will have a contrary result and make the movement slower. This will happen for machines with a very fast controller, that checks the angle between two consecutive line segments on the toolpath and only keeps up full speed in case these segments are almost parallel. Deleting intermediate points will increase the angle between the remaining line segments and force the machine to slow down on every corner. So for these machines you should not check this option.

4.3.7 Postprocessor

The postprocessor determines the format of each NC file to be written. Compare this with a printer driver, which 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 the ISO standard is followed small differences are present), 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 8 or 9 tab pages. It is not always possible to completely display all tabs within the available space. DeskProto will then add two arrows at the right side of the tabs, allowing you to select which tabs you want to see. The image above shows how this looks in Windows.

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, remove and copy postprocessors by adding/ removing/copying PPR files to/from the DeskProto

Drivers folder (as set in the Preferences). The Drivers folder can be found by pressing cancel here and then button 'Open location' in the underlying dialog.

Button **Notes** will open the Notes dialog, in which you can enter extra information that you want to store in this postprocessor definition. Details like when it was created, and who created it.

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 (the first few lines often may not be numbered)
- **Start command** - command to be written before the actual line number (in G-code usually an "N")
- **End command** - command to be written after the actual line number (usually empty - not used)
- **Start value** - the first line-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: 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 value 0 entry all lines are numbered, for value 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). These values are standard ASCII control characters.

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. You can look at an existing postprocessor for examples. Also looking in an existing NC program file that works OK on your machine is helpful here.

The screenshot above shows two lines with start commands and two with end commands (this is just an example). The first line ("%") is the standard start line of many G-code implementations. "G21" tells the machine to use mm as units (the alternative being G20 for inches), and "G90" tells to use absolute coordinate values (the alternative being G91 for relative coordinates).
End command "M5" tells the machine to stop the spindle, and "M30" means end of program.

On the left side of the screenshot you will see that a "**middle dot**" (·) is displayed between G21 and G90, instead of the space that was programmed there. Reason is that it is often difficult to see if a line starts with a space or not: the middle dot makes it very clear where spaces are present. When editing this field (as shown on the right side) and when using Show example you will again see a space.

In both edit boxes you can use **placeholders**. These are special bits of text that you can enter, which will be replaced with a parameter value when the NC program file is written. For instance the text "`{MACHINENAME}`"will be replaced by the name of the machine that is being used. A placeholder that forces a new line in the NC program is available as well: `{N}`. A list of all available placeholders can be found at page Postprocessor placeholders.
For machines that support the use of **comment lines** in the NC code this is of course a great place to enter comment lines, showing information for the machine operator, like when the NC file was generated, the size of the block, which cutter needs to be loaded, etc.

At the bottom of the page a large button **Show example** is present. When you press that button the two large edit boxes will change in two example panes, showing what the result will look like in the NC file. The placeholders will be processed using the settings that DeskProto finds in the first part and first operation of the current project. The button at the same time will change to **Hide example**, that can be used to return to the two edit boxes. So the button in fact toggles between the two situations.

Movement settings

The Tab page Movement determines the format of all movement command lines in your NC program, which will be 99.99 % of its contents.

Each movement (so each line in the NC file) is built using a Start command, some Coordinates and an End command. To see their effect: just look at the **Example** line at the bottom 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 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 positions is set to 7 and the output is 3.000 two spaces will be added, making the result in this example 'X 3.000'. The default is 1, for 'do not add spaces'.
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 is also applied when drawing the toolpaths on the DeskProto screen. As a default for mm 3 decimals are used, and for inch 4.
The calculations of DeskProto are either in mm or inches (whichever you configured in the Preferences). 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. This is used in several NC formats, as it allows to use integers as coordinate values instead of real numbers.

The factor can also be used to reverse the direction of the axis: by entering -1.00 (minus one) 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.
In most cases you can simply leave this field empty, as it is needed only for a few machines that need a non-standard proprietary format.

In all these edit fields a **space** will be shown as a "**middle dot**" (·), like in the example screenshot in front of the X, the Y, the Z, the A and the B. This is done as otherwise spaces would easily be overlooked. When editing the field and in the example line you will again see a space.

In all Start command and End command edit boxes you can use **placeholders**. These are special bits of text that you can enter, which will be replaced with some parameter setting when the NC program file is written. For instance the text "`{TOOLNUMBER}`"will be replaced by the number of the cutter that is being used. A placeholder that forces a new line in the NC program is available as well. A list of all available placeholders can be found at page Postprocessor placeholders (despite the name they can also be used in this dialog).

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.

What you enter as **Separator** will be inserted between the coordinates of the movement. For instance when you enter "," (a comma) the result may look like X10,Y20,Z30. Using a space as separator is possible too. The insertion is done only in-between the coordinate values, so not for instance after the movements Start command or before a Spindlespeed value that is added to that same line.

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 A180*" 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** 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 G20 (inches) or G21 (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 setting has been added to be complete, though in fact we have never yet seen a machine that needs angle values in radians.

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.
Here again the **Example** line shows what will happen for your settings.

Feedrate settings:

Feedrate commands will only be output if the option **Use feedrate command** is switched on.
The option **Write only if changed** will be on for most machines: if not the feedrate will be output on (or for) every movement line.

The Feedrate command is constructed using a **Start-command** a number and an **End-command.** The result can be seen in the example line at the bottom of this tab page.
Positions stands for the minimum number of positions that are occupied by the value of the feedrate, normally this will be 1.
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. This is done when the machine needs to read value 50000 to set a speed of of 50 (mm/sec).

In the Start-command and the End-command you can use **placeholders**. These are special bits of text that you can enter, which will be replaced with a parameter value when the NC program file is written. For instance the text "`{F}`" will be replaced by the current Feedrate. A placeholder that forces a new line in the NC program is available as well, which allows you to construct a command that uses two lines. A list of all available placeholders can be found at page Postprocessor placeholders (despite the name they can also be used in this dialog).

In these two edit fields a **space** will be shown as a "**middle dot**" (·), like in the example screenshot in front of the F. This is done as otherwise spaces would easily be overlooked. When editing this field an don the example line you will again see a space.

The **Method** setting is self-explanatory.

The **Units** need to be set to the feedrate units that your machine needs. In case these are different from the units for the DeskProto user interface (which you have set in the Preferences) DeskProto will convert the value to the correct units. The conversion is new in DeskProto V8 - older versions simply copied the value as entered for an operation to the NC 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 defined 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 linear length of a rotary movement depends on the distance to the rotation axis: the further from this axis, the larger the length (for the same angle). 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. Still some machines expect this type of feedrate.

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 - both extra tab pages - will be explained below.

The **Example** line shows what the result will look like, using the feedrate found in the first operation of the current project.

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 that will be used 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.

For historic reasons the parameter for this option is not the time but the **inverted time** (1 divided by the total time). The resulting F-values in the NC file will seem weird as they can be very high - do not worry as such high values are perfectly normal.

The Time for each movement can easily be calculated by DeskProto, 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, as there the distance to the rotation axis is larger.

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.

The **Example** line shows what the result will look like, using a value of 1.

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 that will be used 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, for every movement that includes an angle. Important is that when roughing layers are applied the (linear) Distance to be machined (needed for this conversion) 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, as there the distance to the rotation axis is larger.

In case your machine also supports 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.

The **Example** line shows what the result will look like, using a value of 1.

Spindle speed settings

Spindle commands will only be output if the option **Use spindlespeed command** is switched on.

The Spindlespeed command is constructed using a **Start-command** a number and an **End-command.** The result can be seen in the example line at the bottom of this tab page.
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 0.01666 will output a number that is 60 times smaller than what DeskProto shows. For instance 167 revolutions per second instead of 10000 revolutions per minute (though we have never seen a machine that requires such conversion).

In the Start-command and the End-command you can use **placeholders**. These are special bits of text that you can enter, which will be replaced with some parameter setting when the NC program file is written. For instance the text "`{S}`" will be replaced by the current Spindlespeed. A placeholder that forces a new line in the NC program is available as well, which allows you to construct a command that uses two lines. A list of all available placeholders can be found at page Postprocessor placeholders (despite the name they can also be used in this dialog).

In these two edit fields a **space** will be shown as a "**middle dot**" (·), like in the example screenshot in front of the S and the M3. This is done as otherwise spaces would easily be overlooked. When editing this field and on the example line you will again see a space.

The option **Apply the Dynamic feedrate control percentages also on the Spindlespeed** is about an option on the Movement tab of the Geometry operation parameters. The **Dynamic feedrate control** options on this tab page allow you to reduce the feedrate in certain circumstances. When the feedrate is reduced the spindlespeed may become too high, so the cutter no longer can cut clean chips. This postprocessor option makes DeskProto reduce the spindlespeed as well, with the same percentage.

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, or Revolutions per Minute).

The **Example** line shows what the result will look like, using the spindle speed found in the first operation of the current project.

Toolchange settings

DeskProto will create one combined NC program file for all operations in a part. However, for every operation a different cutter can be used, so in case you are using various cutters in one part (which would thus mean in one NC program) you have to define what must 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 (Note that this is just an example: probably not valid for your machine). The first line in this example ("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” (spaces are shown as "high dots"). For many G-code machines this first line will be sufficient. 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.

In all edit fields you can use **placeholders**. These are special bits of text that you can enter, which will be replaced with some parameter setting when the NC program file is written. For instance the text "`{TOOLNUMBER}`" will be replaced by the tool number current cutter. A placeholder that forces a new line in the NC program is available as well, so if needed you can use more than three lines. A list of all available placeholders can be found at page Postprocessor placeholders (despite the name they can also be used in this dialog).

In all these edit fields a **space** will be shown as a "**middle dot**" (·), like in the example screenshot in front of the M06. This is done as otherwise spaces would easily be overlooked. When editing this field and in the example line you will again see a space.

Note:
For this method be sure that the ‘Number of tools’ of the machine is set correctly. You can do that in the Machine definition. 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.

Note 2:
Some machines that do not have an Automatic Tool Changer still accept a tool change command. These machines will then start some process that allows you to **manually change** the cutter. For such machines the Number of tools in the Machine definition needs to be set high enough, we usually enter 99.

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 of course 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 this option. 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 toolpaths that need multiple NC program files, 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

The **Example** line shows what the result will look like, using the tool number found in the first operation of the current project.

Laser settings

DeskProto supports machines with a laser engraving option. Such option means that instead of a cutter a laser is used, with its beam pointed downwards (vertically). Where the laser beam hits the surface of the material that will burn, and the resulting discoloration can be used to 'engrave' text and/or graphics. The burning can also be used to cut a path through thin material: laser cutting. DeskProto supports laser engraving / laser cutting only for vector operations.

Laser engraving is possible only when on this page the option **Use** is switched on and when a laser option has been configured in your machine definition. You then can activate it by selecting a laser cutter in the vector operation parameters.

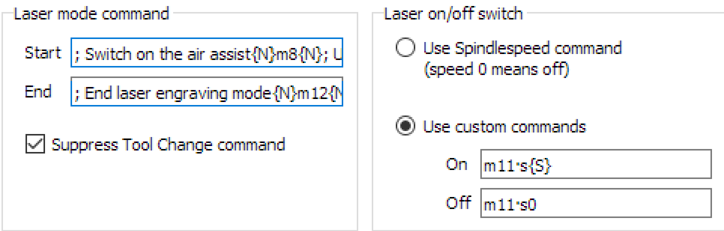
The Laser **mode commands** are the commands to switch the machine to laser mode (**Start**) and to switch back to CNC machining (**End**). In the example above no such command is present, as on that machine laser mode is switched on by loading cutter number 100 and again switched off by selecting a different cutter.
The M8 and M9 in the edit fields in this example are to use the laser's **air assist**: blow air over the area in order to remove the smoke and the debris and to cool the material. Without this air assist the smoke may pollute the laser's lens, and the laser or part may become too hot. M7 and M8 are the most used commands to turn on air assist, M9 to switch all coolant off.

So on the machine laser engraving can be enabled either by selecting a special cutter or by using this special laser mode command. However, in DeskProto laser engraving can be enabled only by choosing a laser cutter (in a Vector operation). When the postprocessor includes a toolchange command DeskProto will write that command, telling to load the laser cutter. Which is not needed for machines that use a laser mode command: for these machines you can check the option **Suppress Tool Change command**.

On many machines the Spindlespeed command ("S") is used as **Laser on/off switch**. For these machines you can simply select the option **Use Spindlespeed command**. Setting the S-value to 0 will switch the laser off, setting a different value will switch at on, at a certain power level. In a laser toolpath DeskProto will automatically set the S-value to 0 for all positioning movements.

When your machine needs different commands to control the laser power you can select **Use custom commands** and enter the required commands for On and Off.
The third option is **None**, which can be used for machines that automatically control the laser power: laser off for rapid moves. Some machines even will lower the laser power when the feedrate drops at sharp corners.

The **Example** line on this tab page shows the two Laser on/off command lines.



This is a second example, with completely different settings, to show that the settings may be very different per machine.
The image above shows the settings created for an Eding CNC controller. These including both commands and comments.

The **Laser mode** Start command for this machine is:

```
; Switch on the air assist{N}m8{N}; Use dynamic power control
```

which will result in 4 lines in the NC file:

```
; Switch on the air assist
m8
; Use dynamic power control, minimum power 10%
m12 s10
```

The lines that start with a semicolon for this controller are comment lines, and {N} is a placeholder that inserts a new line.

The Laser mode End command is:

```
; End laser engraving mode{N}m12{N}; Wait 5 sec before switch
```

with these lines as result:

```
; End laser engraving mode
m12
; Wait 5 sec before switching off the air assist
g4 p5
m9
```

As **Laser on/off switch** his machine uses a custom command: M11. The Start commands needs the S-value, which is inserted using placeholder {S}.

The example line shows both the Laser On and the Laser Off command.

Hot wire settings

DeskProto supports machines with a hot wire cutting option. Such option means that instead of a cutter a hot wire is used: a horizontal wire, parallel to the Y-axis of the machine. This wire is heated and can cut EPS foam by melting it. The wire follows a 2D toolpath, moving along X and Z. DeskProto supports hot wire cutting only for geometry operations and for bitmap operations.

Hot wire cutting is possible only when on this page the option **Use** is switched on and when a hot wire option has been configured in your machine definition. You then can activate it by selecting a hot wire cutter in the operation parameters.

The hot wire **mode commands** are the commands to switch the machine to hot wire mode (**Start**) and to switch back to CNC machining (**End**). DeskProto does not offer an option to set the power or the temperature of the wire; if needed such command can be included in the Start command, with a fixed value.
In the screenshot above no mode commands are present, as on that machine hot wire mode is switched on and off manually.

On the machine hot wire cutting can be enabled either manually or by using this special laser mode command. However, in DeskProto hot wire cutting can be enabled only by selecting a hot wire cutter (in the operation parameters). When the postprocessor includes a toolchange command DeskProto would write that command, telling to load the hot wire cutter. Which is not needed for most hot wire machines: for these machines you need to check the option **Suppress Tool Change command**.

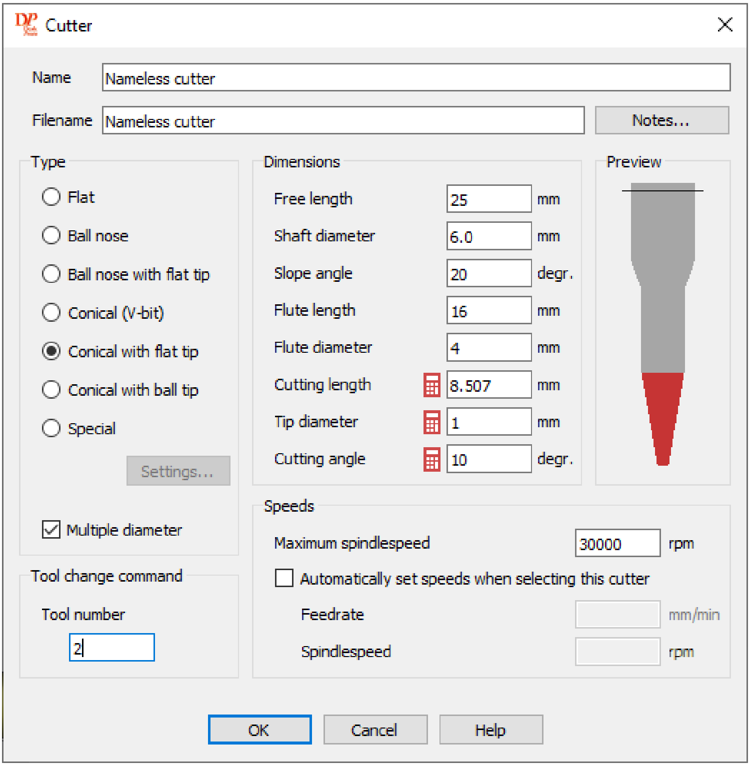
Many hot wire cutters use names for the three **Axes** of the machine that do not match the XYZ convention as used for CNC machining. Hot wire cutting basically is a 2D process, so on many machines only the X and the Y are used. The X axis is the same as for CNC milling, however the Y coordinate on the machine is in fact the Z-coordinate in DeskProto (the height of the hot wire). For these machines DeskProto needs to use the "Y" as **Character for the Z-axis**. We have also seen machines that use character "C" for the height of the hot wire.

The hot wire is attached to the machine on two sides, and advanced hot wire machine support an option to make each side follow a different path. For such complicated "3D" path an NC file is needed with four coordinate values for each G1 command, we have for instance seen machines that use X, Y, U and V coordinates. X and Y for one end of the wire, U and V for the other end.
DeskProto does not offer the option to use different paths for both ends of the wire. When a machine still needs 4 coordinates for a simple 2D path you can configure DeskProto to **Use two slave axes**: for the machine just mentioned you need to enter "U" as slave of X and "V" as slave of Y. The G-code command "*G1 X100.0 Y200.0*" then will be written as "*G1 X100.0 U100.0 Y200.0 V200.0*".

It will be clear that changing coordinate characters in the NC file only is possible when characters are used to indicate the axes. For many postprocessors that is not the case, meaning that axes cannot be renamed in the NC file. Because of this these Axes options are available **only for G-code based postprocessors**.

The **Example** line on this tab page shows what the movement command in G-code will look like with the options that you configured here.

4.3.8 Cutter



The screenshot above shows the definition for a very strange cutter. It has been configured only for this screenshot: in order to have all dimensions activated and also visible in the preview drawing.

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. We advise to 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, remove and copy cutters by adding/removing/copying CTR files to/from the DeskProto Drivers folder (as set in the Preferences). The Drivers folder can be found by pressing cancel here and then button 'Open location' in the underlying dialog.

Button **Notes** will open the Notes dialog, in which you can enter extra information that you want to store in this cutter definition. Details like when it was created, and who created it.

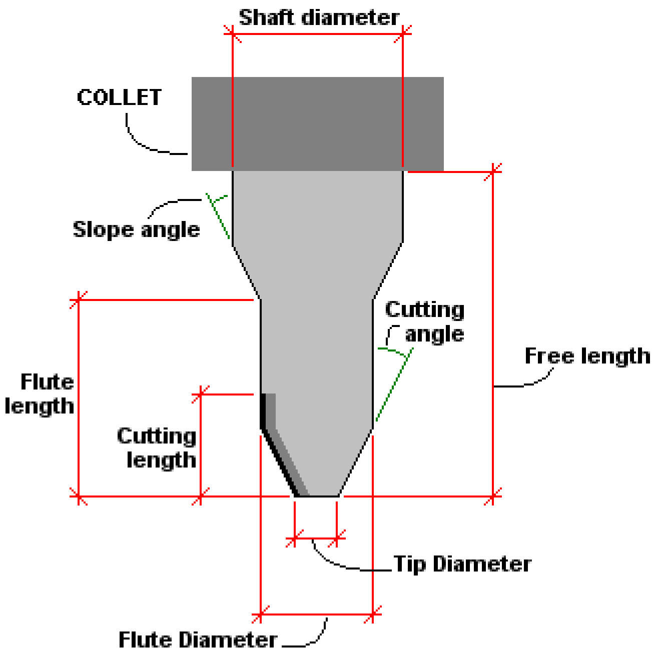
DeskProto offers seven **Type**s of cutter, six of which are variants of three basic types: flat (square cutter), ball nose (spherical tip) and conical (V-shaped cutter or tapered cutter). Both conical and ball nose can be given a flat tool-tip, conical can be given a ball nose tooltip as well. For that last type (Conical with ball tip) check the image on the bottom of this help page for the exact meaning of 'Tip diameter'.
For cutter type **Special** an extra dialog is present, to configure the Special Cutter settings.

Depending on the type that you select, some of the **Dimensions** will be disabled. For example a flat cutter does not have a cutting angle. In order to support 1/8 inch (3.175 mm) cutters with your DeskProto configured to use mm, all dimensions in this dialog can be set with 3 decimals accuracy.

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 (almost) 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.

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. DeskProto needs this information to make sure that the thick shaft and the sloped part will note collide with your part.

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 can be set only 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. For conical cutters with a ball tip see the explanation of this setting at the bottom of this help page.
For Ball nose cutters (also called '*bull-nose* cutters") this setting is named **Corner radius**, as that is the way that Ball nose cutters with a flat tip are defined. For instance a Ball/Flat cutter with diameter 10 and corner radius 2 will have 6 mm diameter flat area at its tip.
A special cutter type can be defined by choosing “Ball nose with flat tip” and setting the Corner radius larger than half the Flute diameter (which normally is nonsense). This creates a special cutter with a curved tip, as shown in the preview drawing. Such cutter also is called a 'Lens type cutter'.

The **Cutting angle** is for conical cutters only: it is the angle between the cutting edge and the center-line of the tool (so the side angle, or ‘grinding angle’). Do not confuse this with the ‘included angle’ that is also used in cutter definitions.
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.

Next to each of the last three edit boxes a **calculator icon** is present, which is activated (color red) for all conical cutters. Clicking this icon will make DeskProto calculate the value of this setting, based on the other two. For instance: the cutting length will be calculated based on the Tip diameter and the Cutting angle, assuming that the cutting edge stops at the end of the angled part of the cutter.

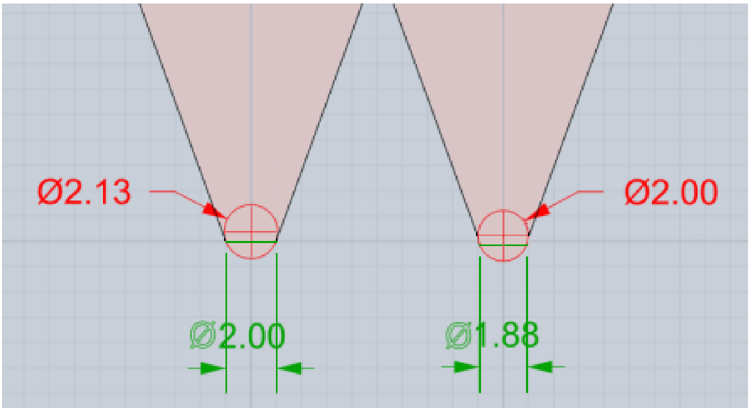
The **Preview** shows you the currently defined cutter-definition in a drawing, which is a very convenient help when setting the parameters in this dialog. A preview can of course only be drawn when a correct set of dimensions have been entered first (for invalid setting a red cross will be shown). The area in dark red shows the cutting length, the horizontal line at top indicates the bottom of the collet and thus shows the free length.

The **Tool number** indicates on which position of the Automatic Tool Changer (ATC) this cutter needs to be loaded. So this is used only when your machine supports a **Tool change command**. 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 of the ATC.

The **Maximum spindlespeed** is used only for validation of your projects (some larger cutters may not rotate at a high rpm as they 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.



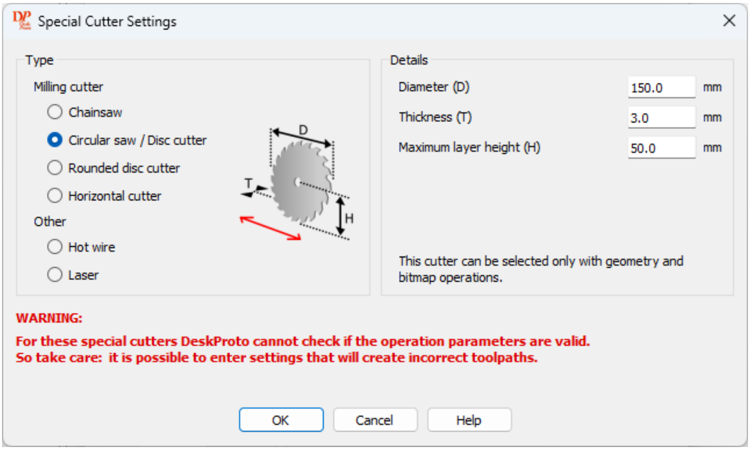
For cutter type **Conical with ball tip** an extra remark is needed, to exactly explain the dimensions of this cutter type. We have found that cutter manufacturers use two different methods to define the **tip diameter**. See the image above, drawn for a conical cutter with a tip diameter of 2 mm.

- on the left: **the 2 mm is the diameter at the tangency point** of angled cutting side and tip radius (the green 2 mm dimension). The conical part stops at 2 mm diameter, and then a ball is added with a diameter that makes it tangent to the cutting side. The diameter of that ball will be a bit larger than 2 mm.
- on the right: **the 2 mm is the diameter of the (half) sphere is used as tip** (the red 2 mm dimension). Now the diameter of the ball is 2 mm, and the

cutting side is made tangent to that ball. The diameter of the tangency point is a bit smaller than 2 mm.

DeskProto uses the first definition (left side of the image): the diameter of the sphere is a (tiny) bit larger than the "tip diameter". The smaller the cutting angle, the smaller the difference.

4.3.9 Special Cutter settings



Most of the cutters in DeskProto (defined in the Cutter definition dialog) rotate round a vertical axis: basically a cylinder shaped cutter that is clamped in the collet and cuts by rotating round the central axis of this cylinder.
In this dialog you can define a few **Special cutters**. You can reach it by pressing button Settings for type Special in the Cutter definition dialog.

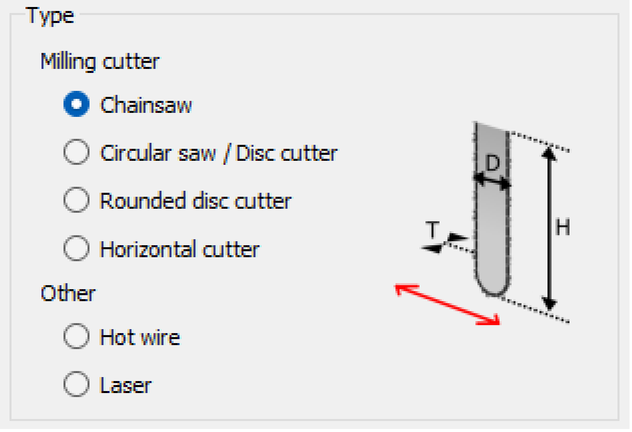
Be careful: DeskProto has been designed for standard cutters. The special cutters in this dialog have been added on user request, for special applications. The number of users is low, and thus these toolpaths have not been tested as extensively as paths for normal cutters. So carefully check your results before starting your machine !

Six special cutter types are available, of which the first four are **milling cutters**: they function by cutting small chips off and thus making the block smaller. The **Other** two cutter types do not make chips, instead they burn or melt the material . These two need a machine that offers this special tool, and therefore such cutters *can be selected only when the tool has been configured*: both in the current machine definition and in the current postprocessor.

Most special cutters also cannot be used in all three operation types (vector, geometry and bitmap).

These special cutters can be very large (compared to the part). As a result it can happen that in the Toolpath animation, when the 'True cutter' is shown, the cutter is not drawn completely as it partially falls outside the 3D Viewport (the 3D space that is rendered).

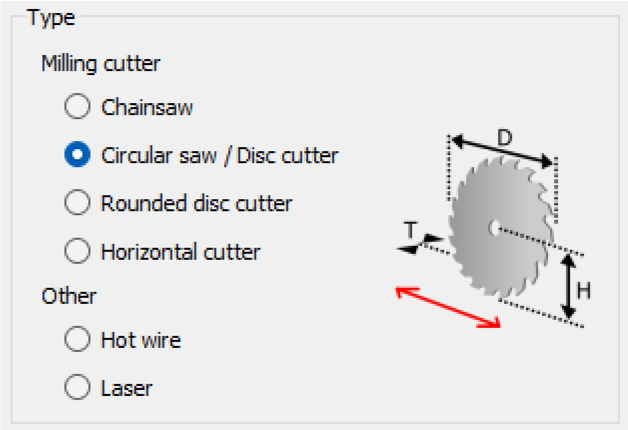
Chainsaw

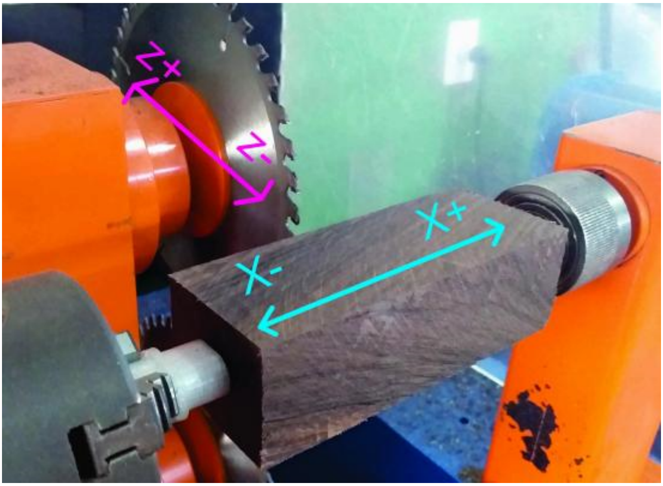


A chainsaw can convert solid wood to chips much quicker than a 'normal' cutter can. So when you want to produce large wooden parts as fast as possible it makes sense to use a chainsaw for roughing.
You can define this special cutter by entering dimensions for its Diameter, Thickness and Maximum layer height. **Diameter** and **Thickness** are explained in the drawing, the **Maximum layer height** tells how deep the chainsaw may sink into solid material (equivalent to the cutting length of a normal cutter).

Selecting this cutter will not only set the shape of the cutter, it will also make sure that inside the material the cutter may move only along a straight path: it may not move sideways and also not change direction. All positioning movements therefore need to be done above the material (a setting called Never Stay Low is applied internally), and as Strategy only Parallel is possible.
Chainsaw cutters therefore cannot be used in vector operations.

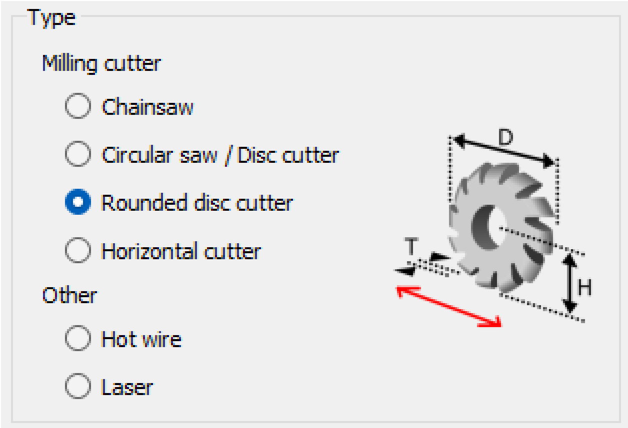
Circlar saw / Disk cutter

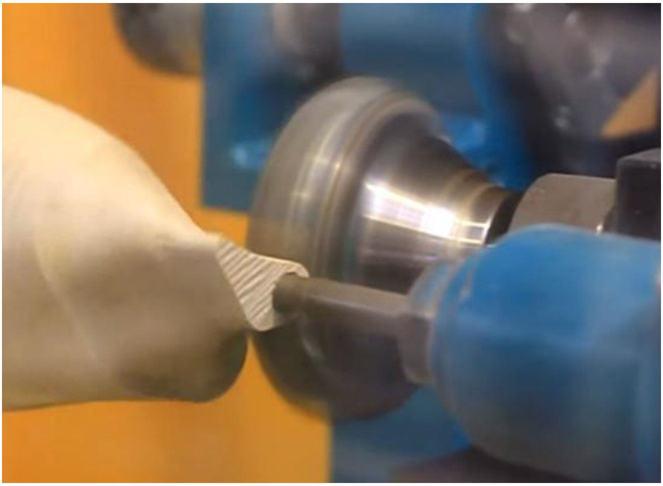




Here the same settings and limitations apply that have been explained for the chainsaw.
Note that this cutter type is **not** about a T-slot cutter or side-milling cutter, with a small horizontal sawblade on a vertical shaft that can be loaded as a normal cutter. This is about a cutter as shown on the photo, which is used for a very fast roughing operation.
Circular saw cutters also cannot be used in vector operations.

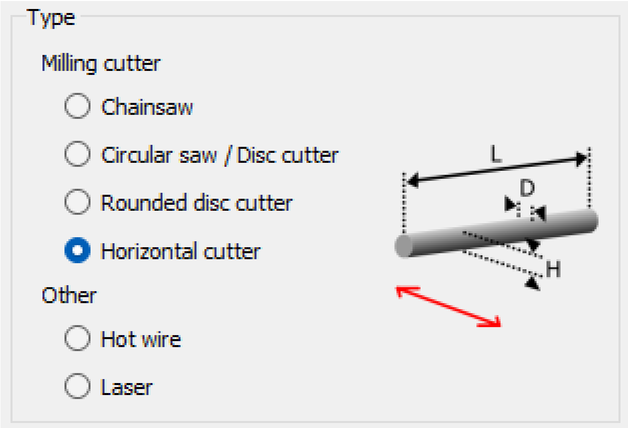
Rounded disk cutter

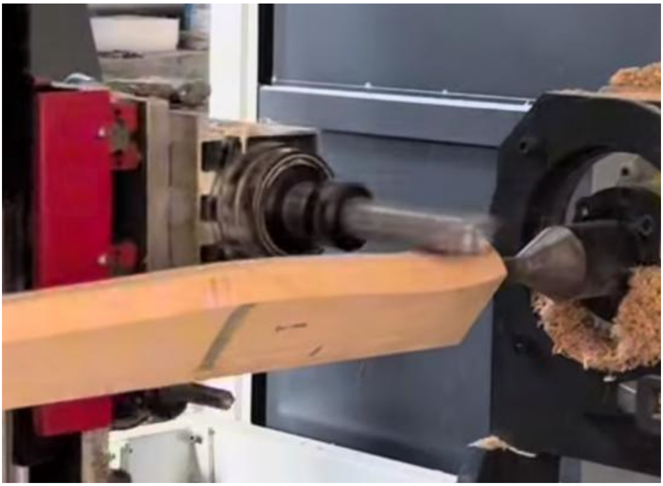




Again the same settings and limitations apply that have been explained explained for the chainsaw.
We have seen this type of cutter in special machines, for instance to produce shoe lasts, as shown in the photo.
Rounded disk cutters also cannot be used in vector operations.

Horizontal cutter





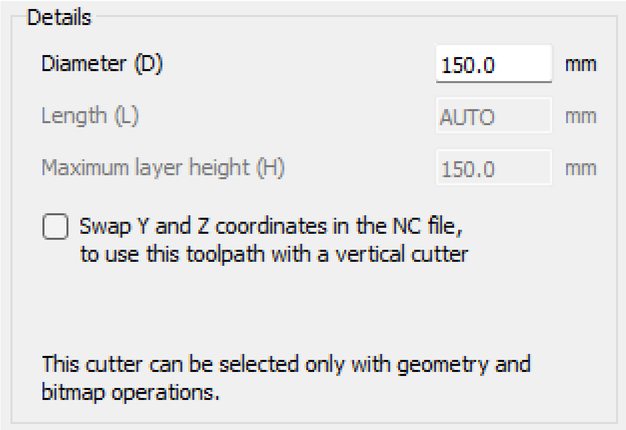
This cutter type will mainly be used in combination with a rotation axis, as a method to create very efficient (roughing) toolpaths. We have seen this on a few machines. You need to make sure that the cutter is long enough as DeskProto does not check that.

The only dimension that you can set for this cutter type is its **Diameter**.
The **Length** of this cutter cannot be set: DeskProto simply assumes that the cutter is long enough to machine the full width of the part in one movement (setting "Auto").
A **Maximum layer height** does not apply either, as for this cutter type roughing layers are not applied

As no roughing layers are applied this provides a very efficient way of roughing, without creating many chips. Though a manual intervention is needed to remove the top part of the block after it has been cut loose. Extra care is needed too: normally after a roughing operation you can immediately start finishing, as all excess material has been removed. This type of roughing does not guarantee that, so some extra roughing may be needed before you can start finishing.

The resulting toolpath in fact is a 2D toolpath: only X and Z are used: for the Y as said the cutter needs to be long enough to machine the complete part. In the resulting NC file DeskProto will automatically suppress this third coordinate. This suppressing does not apply to rotary machining: then the third coordinate is A, and that will surely be used!

Note that the **Z=0** level that you set on the machine for this cutter refers to the **centerline** of the cutter. So you cannot simply set Z=0 at the top of the block by lowering this horizontal cutter till it touches the block.
Horizontal cutters cannot be used in vector operations.

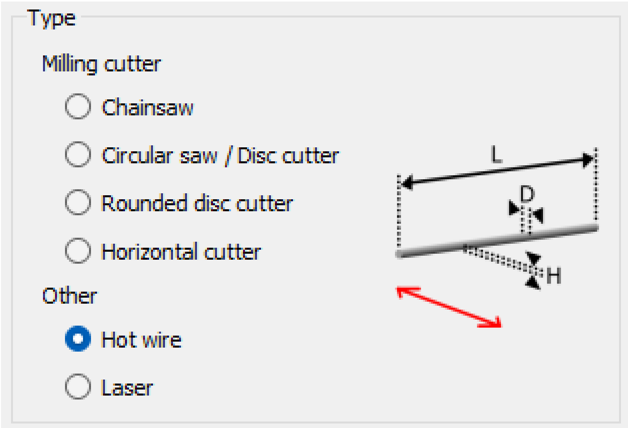


The fourth option that you can select for a horizontal cutter is **Swap Y and Z coordinates in the NC file, to use this toolpath with a vertical cutter**.
As said: this toolpath provides a very efficient roughing method for rotary parts, which can also be useful when your machine only has a standard vertical cutter.That is possible by checking this option: see the photo above. The 2D toolpath now will be an XY toolpath, and the cutter needs to be long enough (and at the correct height for Z=0) to machine the complete part.

The resulting toolpath will be similar to a 'Contour-only' toolpath. However, when combined with rotary machining this cutter will create many such contour paths in one operation.

Note that when we would simply swap Y and Z values the resulting coordinate system would change from right-handed to left-handed, and the machined parts would be mirror images of the parts in CAD. In order to prevent that DeskProto will reverse the sign of all new Z-coordinates (multiply the original Y-value with -1.0) when this option is checked. For rotary toolpaths that won't make a difference as the cutter will remain at Z=0. This will only effect Before- and After-commands in your project. In case this option is combined with a rotation axis parallel to Y (a machine setting) the swapping will be done with X and Z (in which X has already been swapped by the machine setting). Complicated, however it works fine.

Hot wire



The hot-wire is a completely different type of cutter: it cuts a narrow gap through a block of PolyStyrene by melting the material. You can find more information about this technology on page hot wire cutting.

You can select a hot wire cutter only when a hot wire option has been configured for your current machine and in your current postprocessor. As this is about a horizontal hot wire this type of cutter cannot be selected in a vector operation.

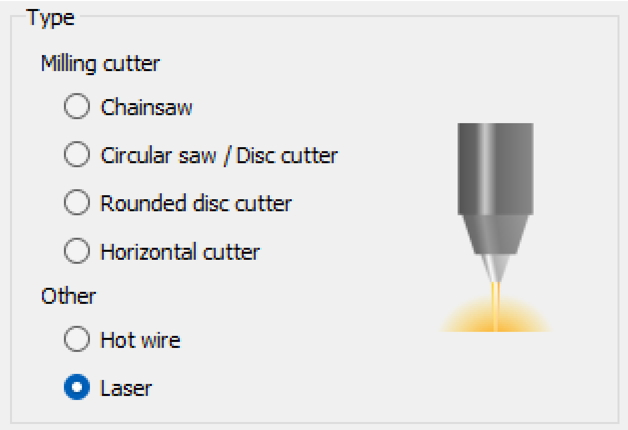
The definition of a hot wire cutter is very similar though to that of the horizontal cutter, see the previous paragraph.
Again the only dimension that you can set for this cutter type is its **Diameter**. Note that this is not about the diameter of the cutting wire, it is about the width of the gap that is created while cutting.

No roughing layers are applied for the hot wire, which provides a very efficient way of roughing: without creating many chips. Though a manual intervention is needed to remove the top part of the block that has been cut loose. Extra care is needed too: normally after a roughing operation you can immediately start finishing, as all excess material has been removed. This type of roughing does not guarantee that, so some extra roughing may be needed before you can start finishing.

The resulting toolpath in fact is a 2D toolpath: only X and Z are used, and both ends of the wire follow the same toolpath. For the Y as said the hot wire needs to be long enough to machine the complete part. In the resulting NC file DeskProto will automatically suppress this third coordinate. This suppressing does not apply to rotary machining: then the third coordinate is A, and that will surely be used!

Note that the **Z=0** level that you set on the machine for this cutter refers to the **centerline** of the wire.

Laser



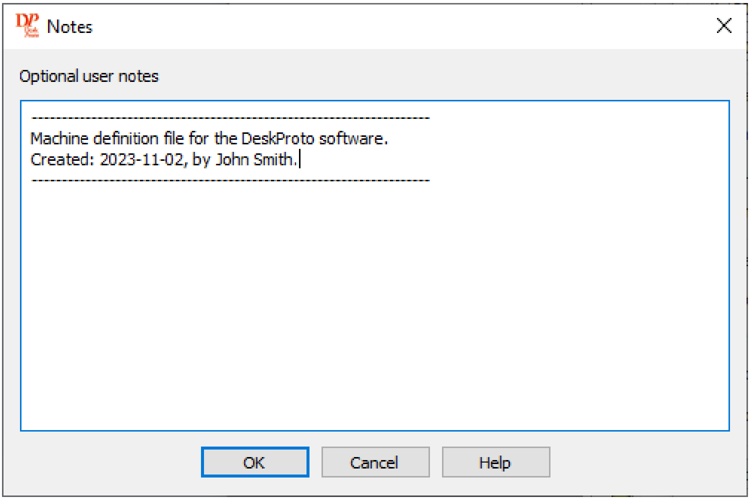
One more cutter that is completely different: the laser cuts by burning the material. It can be used to actually cut through a sheet of thin material or to engrave a graphic image on the material (burning the surface will change the color). In DeskProto it will mainly be used for engraving, as cutting can also be done using a normal cutter. You can find more information about this on page laser engraving.

You can select a laser cutter only when a laser option has been configured for your current machine and in your current postprocessor. As the laser cuts or engraves only at one Z-level this type of cutter can only be selected in vector operations.

The photo above shows a laser cutter applied to engrave on a 3D part that just has been cut on a rotation axis. This is done by selecting the option "Project vector curves on 3D Part geometry", which can be selected only if a geometry operation is present before the vector operation.

The only dimension that you can set for this cutter type is its **Beam width**, also called **Kerf**. The Kerf is the width of the gap that is created when cutting and the width of the line that is drawn when engraving.
The laser power and the laser position can be set in the machine definition.

4.3.10 Driver Notes



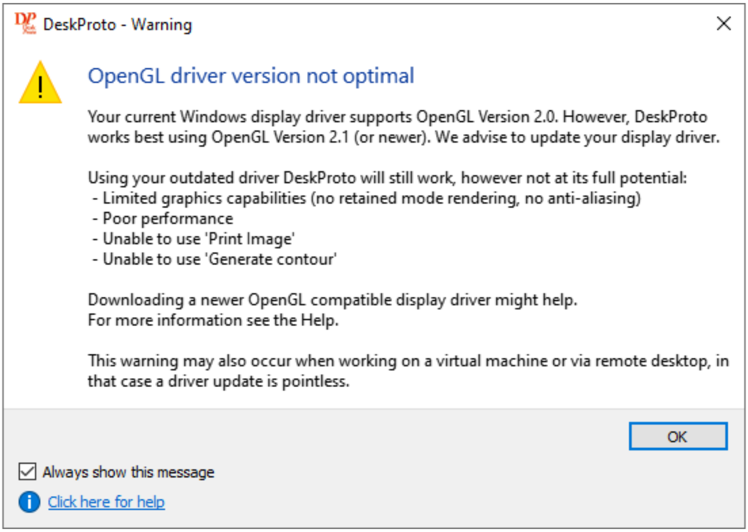
In each of the three dialogs to edit a DeskProto driver (**Cutter**, **Machine** and **Postprocessor**) you can find the button **Notes**, that will open this dialog. The Notes that you enter here will be stored as comment lines at the start of the Driver file that you are editing.

These notes will not be used by DeskProto, they are just for your own convenience. You can for instance store for a certain cutter the exact type and where or when you have bought it. Or any other detail about this driver that is not relevant for DeskProto but that you still want to remember.
You can also use this space for notes to store a "revision history" for the file: when has it been changed, by whom and why.

In the driver file (*.ctr, *.mch or *.ppr) each of the lines will be preceded by a semicolon, which marks them as comment lines.

4.4 OpenGL warnings

4.4.1 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 2.1** or newer, when your driver version is too low this warning will be shown when the program is started.

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.

When this warning is given 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. When your PC and your Windows version are very old this may no loner be possible.

In case you have a choice, select the driver that supports OpenGL.

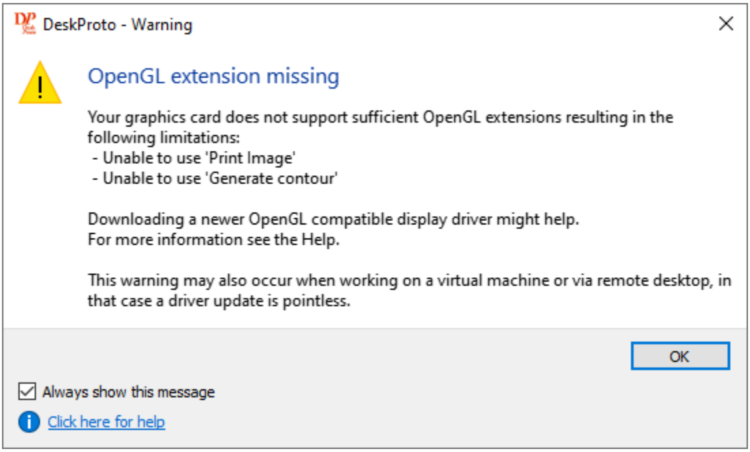
Like for all AMD Graphics cards (formerly ATI), where you can choose between downloading
1- the Catalyst Software Suite
2- the WDM Integrated Driver
Here you need to select the Catalyst suite, as the obsolete WDM driver (Windows Driver Model) does not support OpenGL. Windows then will fall back on its 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 settings can be found in the Preferences. In case of a limited driver not all functions will be available.

4.4.2 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 V2.1 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)
- Generate Contour (Set freeform area).

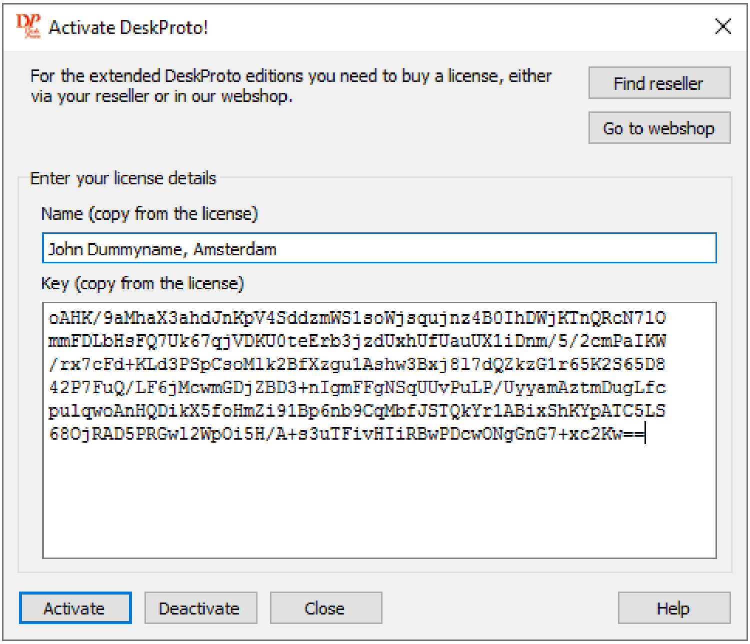
A similar warning will be shown when one of these functions is called, without the list of limitations.

Some other features will simply be disabled (grayed out) when the required extension is missing.
For instance Anti-aliasing (Advanced Preferences).

For **Windows** and for **Linux** is may be possible to solve this by downloading and installing 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. When your PC and your Windows version are very old this may no loner be possible.

For Apple the situation is different: **MacOS** simply does not support the extensions that are needed, so installing an upgrade will not fix this.

4.5 Activate license



In Windows and in Linux this command can be found in the Help menu, in MacOS in the DeskProto menu.

The **Free** Edition of DeskProto can and 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**.

In case you are not sure which edition you need: one of the options in the Free edition is to evaluate (trial) these higher Editions - when trialling a watermark (the "Trial cross") will be visible in all machined parts. On the DeskProto website you can also find a comparison table for these four editions.

In all cases you need to first download the DeskProto installation file from our website www.deskproto.com and instal DeskProto on your computer. This installation file (a Setup file for Windows, a DMG file for MacOS or an AppImage file for Linux) 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, as log as you like.

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 your 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. Even the smallest 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** your license: you will again need Name and Key when (for instance) you buy a new computer.

A registered version of DeskProto will clearly display 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. Needed for instance when you sell your computer.
You can 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 (Windows).

In **Windows** the license activation affects all users of that PC, in **MacOS** and in **Linux** the activation information can be accessed only by the current user: every next user will also need to again activate the license on that same computer.

4.6 Edition select

DeskProto is offered to you as '**Freemium**' software:
you are welcome to use the *basic functionality* DeskProto **free** of charge in the Free edition, the *advanced features* are available as **premium** extras in the three paid editions.

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 can be used as free CAM software, as long as you like. As said: it offers only basic CAM functionality. It also allows you to **Trial** (evaluate) the higher editions; when running in **trial mode** the resulting

toolpath will add a **Trial cross** (watermark), visible 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.
Compared to the Free edition a few important extra options are present: Pocketing, Drilling and V-Carving for Vector operations, and Roughing for all operation types. Projects also may contain any number of parts and operations.

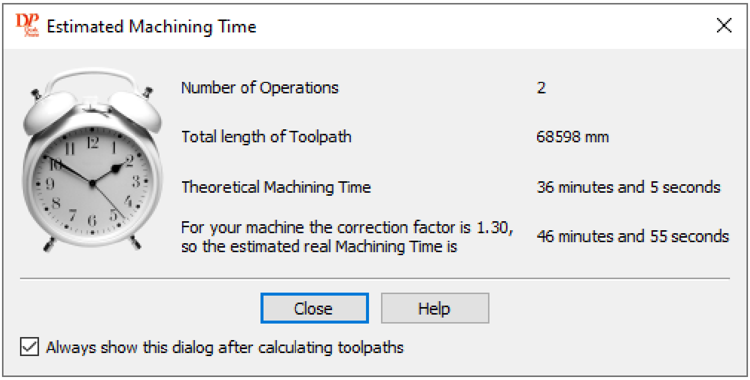
The **Expert Edition** includes all parameters except for the rotation axis options. So the fourth axis and the fifth axis are not available in this edition.

The **Multi-Axis Edition** is the most complete version: all parameters are present, including support for 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 paid edition can be activated in the Activate dialog.
After activating your license this Edition select dialog will no longer be displayed.

4.7 Estimate Machining Time



The **Estimate machining time** dialog will be shown after calculating toolpaths (unless the Always Show option has been un-checked) and it can also be opened via the Create menu. With this option you get a rough estimation of 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 it is easy to calculate the exact machining time: DeskProto knows both the length of the toolpath and the feedrate (machining speed), so length divided by feedrate results in an exact theoretical machining time.
However, the real machining time is influenced by many factors:

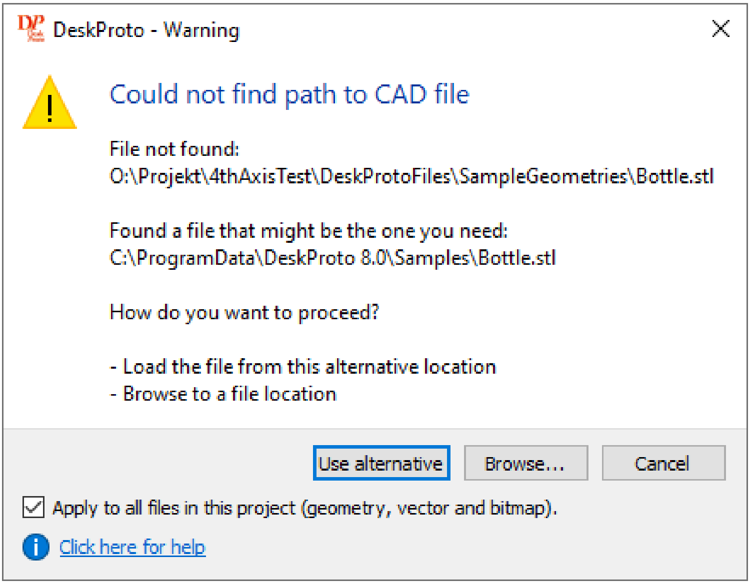
- How long does the machine’s controller need to calculate for one Movement ? (for every linear interpolation it is needed to calculate the speed for each separate axis)
- Does the machine keep up its speed (look-ahead buffer), or does it stop or slow-down in-between each two movements ?
- How fast can 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).

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 Advanced Machine settings. 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.

4.8 File not found warning



When opening a project file, the CAD files that were used will be loaded again: the DPJ file does not store the data but only stores 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.

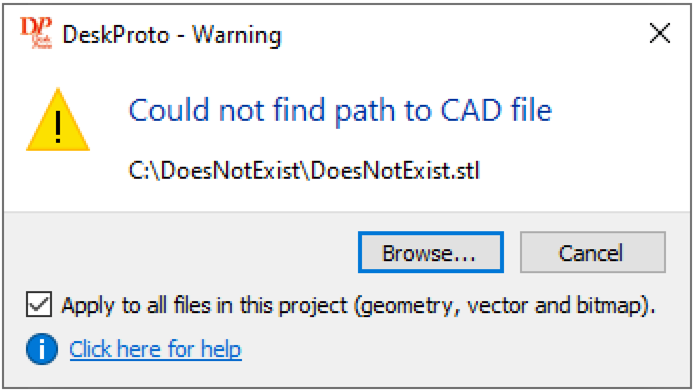
If the CAD file cannot be found (for instance when the DPJ file was copied from a different computer system) DeskProto will check if a CAD file with the correct name can be found in:

1. the current folder (so the same folder as the DPJ file)
2. the default Data folder
3. the Samples folder

If yes, DeskProto will ask you if it can load that file instead, using a dialog as shown above.
You then can choose to either **Load the alternative file** or **Browse to find the 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 new 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.

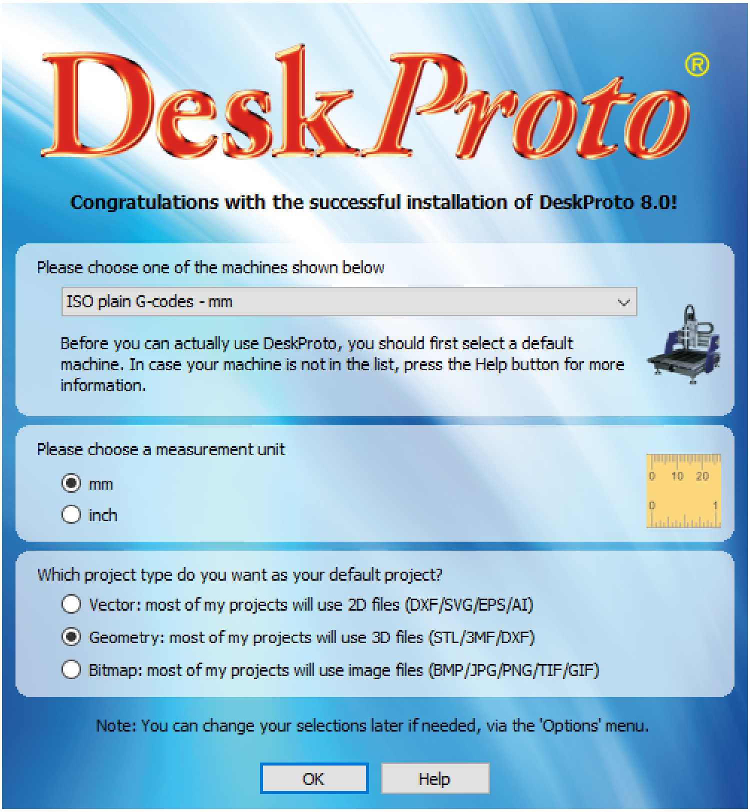
If the file cannot be found on any of these alternative locations this warning-message will be displayed instead:



Here as well you can Browse to find the file on some other location.

After pressing the Cancel button the project will continue loading, without the missing CAD file(s).

4.9 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. The machines are alphabetically listed by their manufacturer's name. 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 default machine that you select here can be changed later, by editing the Default Project (Options menu).

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 are not specified). If needed you can later change this setting in the Preferences.

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 the 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 an option here tells DeskProto which type of project to open as default New project. The other two types remain available as well; simply 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 manufacturers and the models that they produce is almost infinite), and in most cases it is easy to solve. Please follow the four 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 when that works well you can later Add your machine to the list - see below.
Important: when you look for your machine in the alphabetical list you 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 the control software that you use. Like for the Mach3 control software you can use 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 when that works well you can later Add your machine to the list - see 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 when that works well you can later Add your machine to the list - see below.

4. In case the above three options do not work: in DeskProto it is easy to edit the Postprocessor, to make DeskProto write the NC format that you need.

When 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.

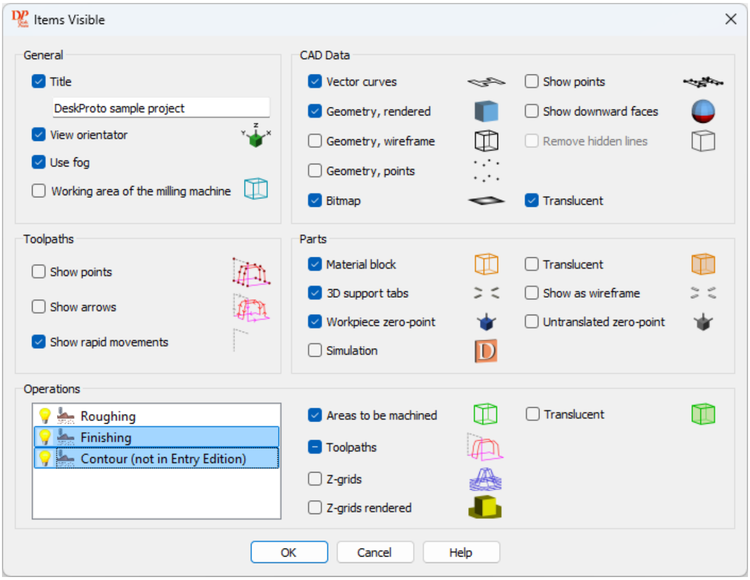
When your machine that was already on the list has an optional rotation axis, it is possible that the machine-definition in DeskProto does not have that option checked. It is easy to configure this in the Machine definition, on the Rotary settings page.

Adding a machine.
When your test (as just described) worked out well 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 definition give the new copy a proper name and filename. Make sure that you do not change the selected postprocessor. Enter the correct dimensions and speeds, if needed add a rotation axis in the Rotary settings. For more information use the Help button of the Machine dialog.

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 there 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.

4.10 Items Visible



In this dialog you can select which items will be included in the View Window.
It can be opened in the View menu.

Shortcuts:

Toolbar: 
Mouse: Double-click in a view, or Right-click and select "Items visible" in the shown context-menu.

Five groups of items are present: General items, types of CAD-data, Toolpath items, Part items and Operation items. In front of each item you find a checkbox: checked ("V") means Visible, not checked means not visible.

Four **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 box 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. Note that 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.

Checking the option **Use fog** applies fog to all drawings that use 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 may have no effect.

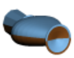
The **Working Area** of the milling machine can be drawn, which is the working area of the machine selected for this project. Of course DeskProto does not know where you will set the workpiece zero point on the machine, so it will draw the part exactly in the middle of the working area. This will give a good indication how the part relates to the machine. The size of 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 curve 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.
The **starting point** of each curve is shown by a larger dot - unless for closed curves as these do not have a starting point.

In a **Rendered geometry** drawing all triangles of the geometry definition are filled with color and then shaded. This offers a good understanding of your geometry, so it is the default way to draw all Geometry data.

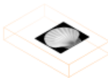


The sub-option **Show downward faces** will assign a different color to any triangles that are facing down (in other words: for which the normal has a negative Z-component). This option makes it very easy to check for Undercuts (areas where the cutter cannot reach), and to optimally rotate your geometry 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 make the result 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).

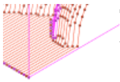
In a **Points geometry** only the three vertices (corner points) of each triangle are drawn.



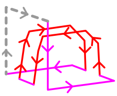
Bitmap concerns the Bitmap image that has been loaded. For large bitmaps DeskProto will display a simplified version (in most cases 512x512 pixels) of the bitmap, in order to speed up drawing.

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.

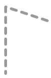
The group that includes the **Toolpath**-related items is the smallest:



The toolpaths that you calculated are shown as red lines. Each toolpath consists of a large number of small straight line segments (linear interpolations). The option **Show points** lets DeskProto draw a small red dot on every point (in the icon drawn in black to make it easier to see them).

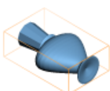


The option **Show arrows** also adds information to the red toolpath lines: the arrows indicate the direction of the cutter movement.

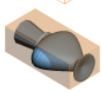


In some situations a very large number of positioning movements over the part is needed, done in Rapid. In such cases is may be difficult to see the 'real' toolpaths as these are obscured by the dashed gray lines of these rapid paths. The **Show rapid movements** option then can be unchecked in order to hide them.

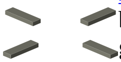
These are the **Parts**-related items:



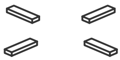
The **Material block** is drawn in orange lines. For standard three-axis machining this is a rectangular block, for rotation axis machining the block may also be a cylinder.



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.



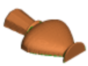
The **3D support tabs** are a special type of geometry, set in the Part parameters, to keep the part connected to the rest of the block. When you check this item they will be drawn, as rendered geometry with a different color (gray).



This is a sub-option for the 3D support tabs, that will **Show them as wireframe** (instead of rendered).

The **Workpiece zero-point** orientator again is a small cube with three axes, this time 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 by the Translation settings 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 **Untranslated zero-point** draws a gray orientator cube with three axes without the part translation mentioned above. This is on the point where in the original CAD-data the zero point is located - unless the option 'Center geometry' has been checked or the CAD data have been translated on Project level, in the Project parameters.

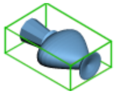


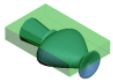
The **Simulation** checkbox allows you to add a Simulation of this Part, which will clearly show what the resulting model will look like. Checking this box will display the complete block of material and show the Simulated Operations dialog. Here all operations for this part are shown, and you can choose the operations to be processed in the simulation.

Finally the **Operations**-related items:
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. When you have selected two operations and an item is checked for one operation and not checked for the other, the checkbox will not show a "V" but a small black square (Windows 10, Linux) or a hyphen (Windows 11, MacOS) like in the illustration above for the Toolpaths.

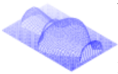
In the list of **Operations** you can change the visibility of each Operation by clicking on the lamp icons (just as in the Project tree). Yellow (lamp on) means visible, gray (lamp off) means invisible.
In the list you can also *Select* one or more Operations by clicking on their line. This will make its background color light blue or gray (meaning selected). Two or more operations can be selected by keeping the Control or

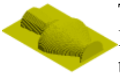
the Shift button depressed when clicking. The Operation Items that you then check and uncheck will apply ONLY to the selected operations.


When checked the **Areas to be machines** will be shown: the rectangular bounding box of each operation area in light green lines.


The sub-option **Translucent** draws the sides of the area in a translucent green color. This will show the area more clearly, maintaining visibility of the geometry inside.

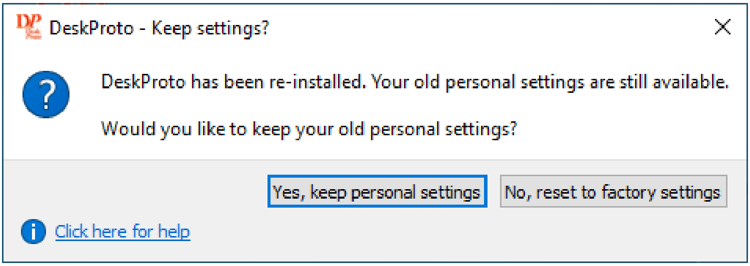

When you check the **Toolpaths** checkbox the Toolpaths of the visible operations will be shown. The toolpath that is drawn is the same toolpath that will be sent to the machine: if there are any problems it should be possible to detect them now.


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.

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.

4.11 Keep settings



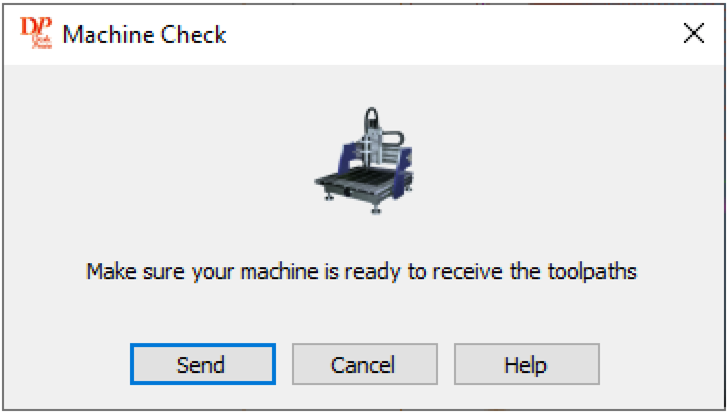
DeskProto stores many of the user's defaults and preferences in the storage facility that your operating system offers(the Registry for Windows, the PLIST file for MacOS and the CONF file fie Linux). When you again install DeskProto you can choose whether or not you want to keep using these settings.

This **Keep Settings** dialog will be shown when have reinstalled **the same version** (for instance you have installed a (newer) DeskProto V 8.0 over an existing DeskProto V 8.0).

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.

A different dialog will be shown when you have reinstalled **a newer version** of DeskProto next to an old one. For instance when you have installed a DeskProto V 8.0 on a PC where a DeskProto V 7.1 was already present.
Then DeskProto the Migrate Settings dialog will be shown.

4.12 Machine Check

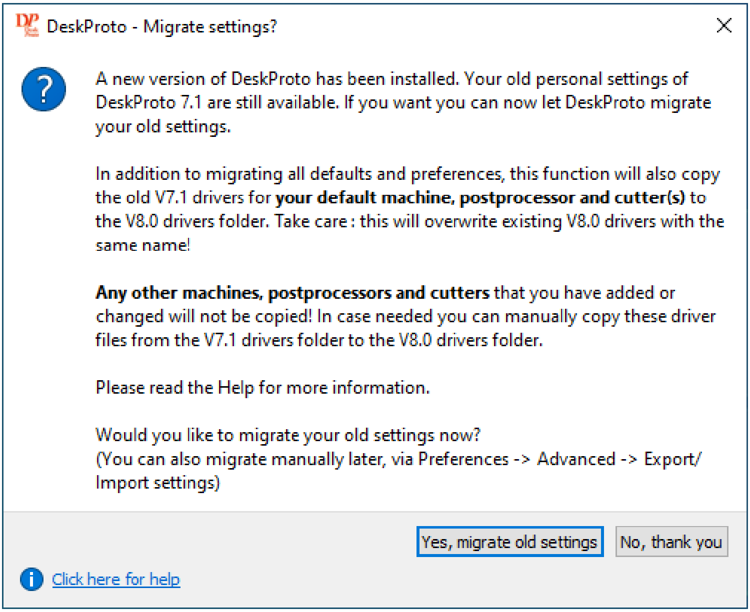


This dialog will be shown by commands Send NC-program file to machine and Send toolpaths to machine, both in menu Create (the first in group Extra).

When a custom icon image has been configured for button "Send toolpaths to machine" (see the explanation on page Preferences) that image will be used in this dialog as well.

The dialog asks 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.

4.13 Migrate Settings



DeskProto stores many of the user's defaults and preferences in the storage facility that your operating system offers:
for Windows this is the Registry, conform Microsoft's guidelines.
for MacOS this is a .PLIST file in the Library (*~/Library/Preferences/com.delft-spline-systems.DeskProto.plist)*
for Linux this is a CONF file (*~/.config/Delft Spline Systems/DeskProto.conf)*
The settings are stored per DeskProto version, so settings for V7.0 and V7.1 are completely independent.

When you have just installed a higher **DeskProto version** on a PC where an older version is present and for the first time start the new version, the above dialog will be shown. 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. For instance you have installed a new DeskProto V 8.0 on a PC where a DeskProto V 7.1 was already present, like in the image above (for you it may show a different old version). This concerns all settings that you made in the Preferences and in the default Project, default Part and default Operation(s). The exact text in the dialog may be a bit different for you: it depends on which previous version of DeskProto has been found)

When you have reinstalled the same version (for instance a new DeskProto V 8.0 over an existing DeskProto V 8.0) the Keep Settings dialog will be shown.

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 V8.0 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 V8.0 drivers folder, this will replace V8.0 driver files with the same name that you just installed! The original V8.0 driver will be renamed to name.bak1 (or name.bak2, etc)
Other machines, postprocessors and cutters that you (may) have added are not migrated. You can manually copy these driver files, see below.

The **Language** setting will not be migrated as the language might not be available in the new DeskProto. If needed you can manually change that in the Preferences.

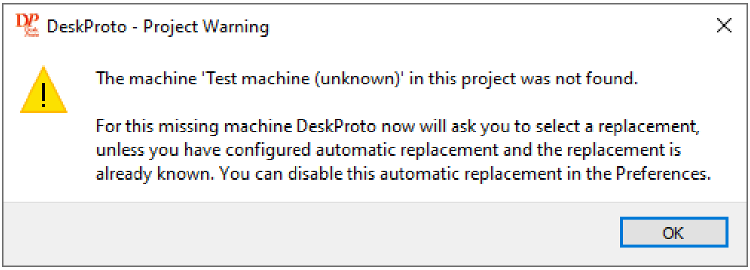
When you answer **No thank you**, nothing will happen, and the new DeskProto will start with system default values.

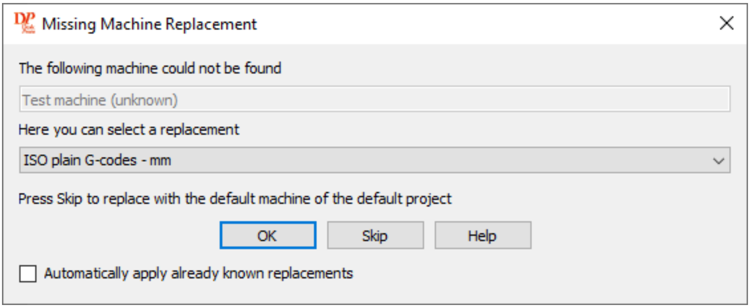
Manually migrating is possible as well:

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 V8.0 Drivers folder. Do not copy all files, as that may remove improvements were made in DeskProto V8. 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. This will only work to migrating settings from DeskProto V6.1 or newer: in older DeskProto versions the Export option will write a .REG file.

4.14 Missing machine





DeskProto stores each machine definition as a .MCH file, in its Drivers folder on the local computer. When you open an existing project on a different computer it may happen that the project needs a machine definition that is not present on that new computer.
In such case DeskProto will first show you the 'Machine not found' warning, and then a dialog that allows you to select a different machine to be used. That is very handy in case (for instance) the name of the machine on this second computer has been spelled a bit differently and thus is not recognized by DeskProto.

When you select a replacement machine and press **OK**, that replacement machine becomes the machine that is set for the project

When you press **Skip** the replacement machine as shown is not used, instead the machine that you have set as default machine in the default project parameters is used.

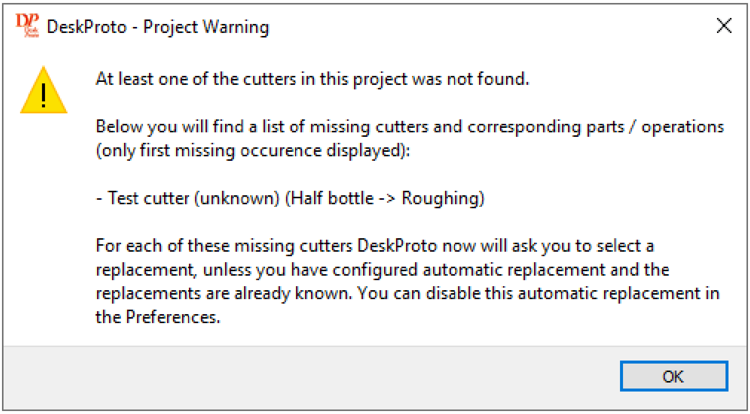
It seems likely that the same replacement will be needed for other projects as well, so DeskProto offers an option to replace automatically: the option **Autmatically apply already known replacements**. When checked, DeskProto will remember the replacement machine that you select here, and

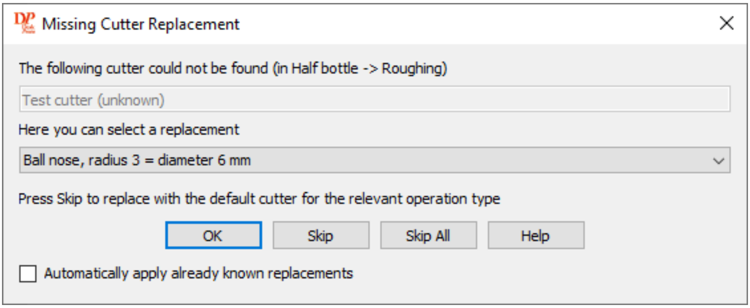
will automatically make that same replacement for projects to be opened later (that need the same unknown machine).
You can again switch that option off in the Preferences: DeskProto will then also forget all "*unknown machine - replacement machine*" combinations that you have set.

The "*unknown machine - replacement machine*" combinations are not copied when exporting and/or importing the DeskProto settings.

The default drivers folder is:
C:\ProgramData\DeskProto 8.0\Drivers\   (Windows)
~/Library/Application Support/Delft Spline Systems/DeskProto/8.0/Drivers/ (MacOS)
~/.local/share/Delft Spline Systems/DeskProto/8.0/Drivers/   (Linux)
You can use that information to copy driver files from one computer to a second one.

4.15 Missing cutter





DeskProto stores each cutter definition as a .CTR file, in its Drivers folder on the local computer. When you open an existing project on a different computer it may happen that the project needs a cutter definition that is not present on that new computer.
In such case DeskProto will first show you the 'Cutter not found' warning, and then a dialog that allows you to select a different cutter to be used. That is very handy in case (for instance) the name of the cutter on this second computer has been spelled a bit differently and thus is not recognized by DeskProto.

When you select a replacement cutter and press **OK**, that replacement cutter becomes the cutter that is set for the project

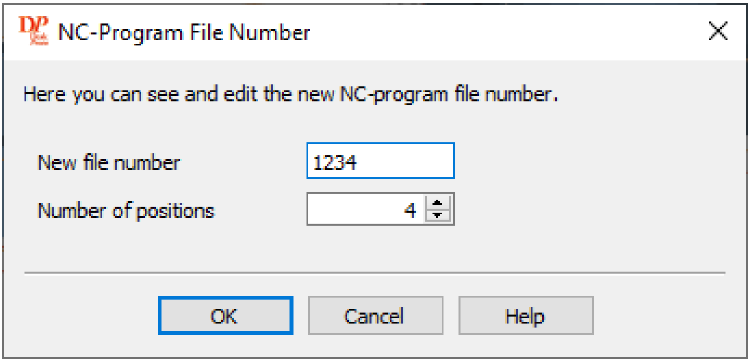
When you press **Skip** the replacement cutter as shown is not used, instead the cutter that you have set as default cutter in the default operation parameters is used.

It seems likely that the same replacement will be needed for other projects as well, so DeskProto offers an option to replace automatically: the option **Autmatically apply already known replacements**. When checked, DeskProto will remember the replacement cutter that you select here, and will automatically make that same replacement for projects to be opened later (that need the same unknown cutter).
You can again switch that option off in the Preferences: DeskProto will then also forget all "*unknown cutter - replacement cutter*" combinations that you have set.

The "*unknown cutter - replacement cutter*" combinations are not copied when exporting and/or importing the DeskProto settings.

The default drivers folder is:
C:\ProgramData\DeskProto 8.0\Drivers\   (Windows)
~/Library/Application Support/Delft Spline Systems/DeskProto/8.0/Drivers/ (MacOS)
~/.local/share/Delft Spline Systems/DeskProto/8.0/Drivers/   (Linux)
You can use that information to copy driver files from one computer to a second one.

4.16 NC-Program File Number

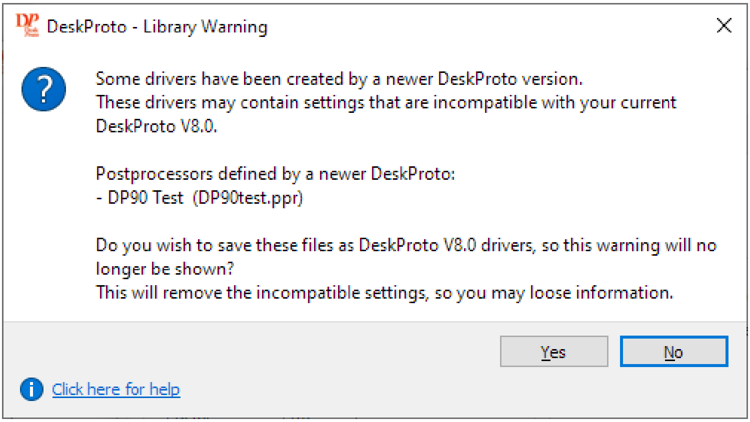


This dialog pops up when you write an NC-Program file, and when you have used the placeholder {PROGNUMBER} in the Start commands of the postprocessor that is used.

It is meant for machines that require each NC-program to have a unique Program Number. Here you can enter the program number to be used in this NC file. On other machines adding this number may generate an error on the machine. Consult your machine's manual to find if this is needed for your machine.

The number of positions sets how many positions will be used: in most cases this needs to be 4 or 5. This information should also be given in your machine's manual. The file number may of course not be larger than possible for this number of positions. When a program number is lower DeskProto will automatically add preceding zeroes. Finally: after writing an NC file DeskProto will automatically increment this number for the next file to be written.

4.17 Newer Drivers warning



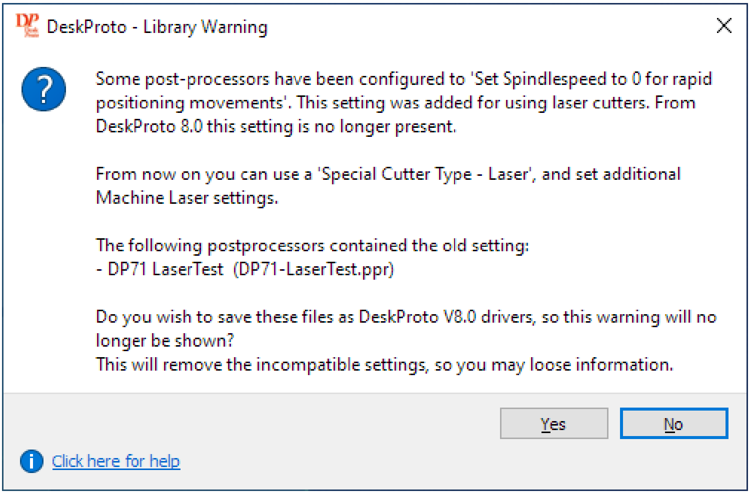
This warning is given at program start, when DeskProto has detected that one of the Driver files (each file contains a definition for a Cutter, a Postprocessor or a Machine) has been created by a newer version of DeskProto.
For instance: your DeskProto V8.0 opens a cutter file that was created by a DeskProto V9.0 (which at the time that this document is written does not yet exist).

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.
An example: DeskProto V7.1 supports "placeholders", allowing you to add information like program number, part size, date, etc to the NC file. These placeholders were not yet available DeskProto V7.0. Using a V7.1 postprocessor with placeholders in V7.0 makes the resulting NC file useless, as in V7.0 the placeholders are not processed.

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 V8.0 files. Any features in the file that are not compatible with DeskProto V8.0 will be deleted.

4.18 Old Laser Postprocessor warning



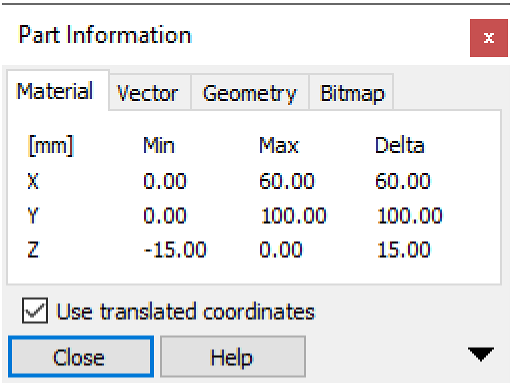
This warning is given at program start, when DeskProto has detected that in one of the postprocessor driver files (files that contain the definition for a Postprocessor) the setting "*Set Spindlespeed to 0 for rapid positioning movements*" has been checked.

That option was available in DeskProto V7.1, on the Spindlespeeds tab page of the Postprocessor definition. It was added to support a very basic form of laser engraving with DeskProto V7.1

As DeskProto V8 fully supports laser engraving this setting is no longer needed (DeskProto now automatically sets the spindlespeed to zero when needed).

This warning will be shown each time that you start DeskProto, unless you select Yes:
in that case these drivers files will be saved again, now as DeskProto V8 files, so without this obsolete setting.

4.19 Part Information



You can find the command to show this dialog in the View menu.
It is a special dialog, as it can remain open while you are working, using other dialogs.

Shortcuts:

Toolbar: 
Again pressing this button will close the dialog.

Depending on which types of CAD data have been loaded in the project, one to four tab pages are shown.
Each tab shows the Boundaries and the 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 are the dimensions of the part that will be created on your milling machine.
When checkbox **Use translated coordinates** has been checked the final Part translation (as set on tab page Zero-point) will be applied as well, so the min and max values then are the NC file coordinates.

You can find the Min and Max values as present in the original CAD-data using button **File info** in the Project parameters.

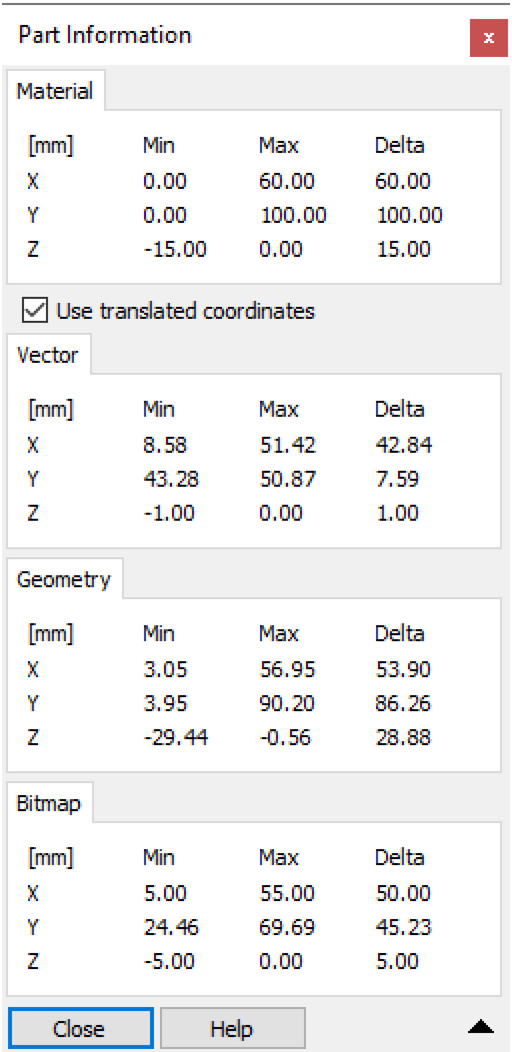
Material:
The dimensions of the material block that has been defined for this part. This is important information for you, as these are the dimensions of the Block of material that you have to prepare.
Vector:

The bounding box of all Vector curves that have been loaded. The machining depth that has been set in the Vector operations for this part (only for the selected curves) is added to the Z-values of the curves. 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. For the Z the relief depth is used that has been set in the Part parameters 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.

For rotation axis projects with a **cylindrical block** of material the Material tab of this dialog shows values for X and D (instead of X, Y, Z). Here D is the diameter of the block (only shown in the Delta column).
In case a **tube shaped block** has been defined the material tab shows a third line, with R values (Radius): the delta between Rmin and Rmax is the material thickness of the tube.

The dialog can be viewed in two ways; as a 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
x
Material
[mm] Min Max Delta
X 0.00 60.00 60.00
Y 0.00 100.00 100.00
Z -15.00 0.00 15.00
Use translated coordinates
Vector
[mm] Min Max Delta
X 8.58 51.42 42.84
Y 43.28 50.87 7.59
Z -1.00 0.00 1.00
Geometry
[mm] Min Max Delta
X 3.05 56.95 53.90
Y 3.95 90.20 86.26
Z -29.44 -0.56 28.88
Bitmap
[mm] Min Max Delta
X 5.00 55.00 50.00
Y 24.46 69.69 45.23
Z -5.00 0.00 5.00
Close
Help

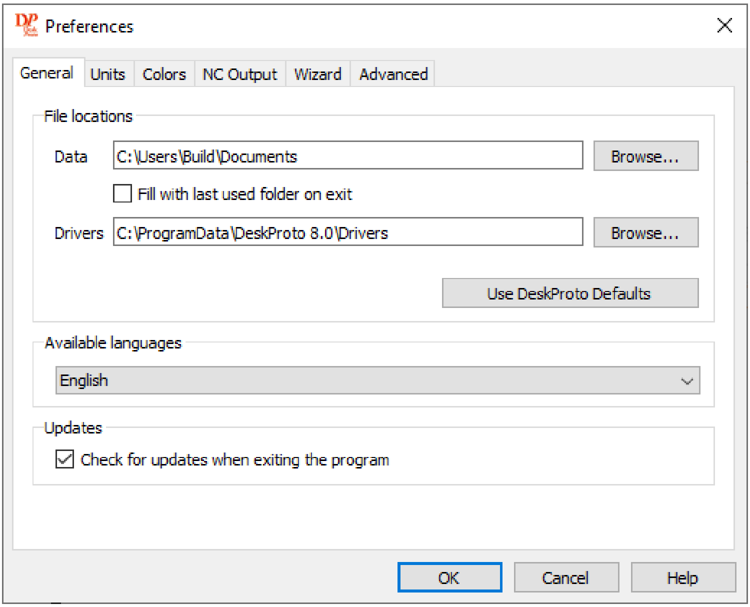
4.20 Preferences

This is a dialog in which you can edit DeskProto's preferences. The dialog consists of six Tab pages. In Windows and in Linux this dialog can be found in the Options menu, in MacOS in the DeskProto menu.

In **Windows** these preferences are stored in the Registry
in **MacOS** in file *~/Library/Preferences/com.delft-spline-systems.DeskProto.plist*
in **Linux** in file *~/.config/Delft Spline Systems/DeskProto.conf*
Each user has his/her own set of preferences stored there.

Note that in MacOS a file called *~/Library/Preferences/com.delft-spline-systems.DeskProto80.plist* is present as well. That is a different file, automatically created by MacOS to remember the settings for the Open file dialog and the Save file dialog.

General tab page



The **File-locations** listed here are the folders (directories) where DeskProto looks for the specified file types. These folders can be changed by typing the new folder's complete path-specification (i.e.: C:\Users\Smith\DeskProtoFiles) 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. This is a standard location in all tree operating systems:
Windows: Documents
MacOS: ~/Documents/
Linux: ~/Documents/
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" on this screen). The Sample file location is:
Windows: C:\ProgramData\DeskProto 8.0\Samples\
MacOS: ~/Library/Application Support/Delft Spline Systems/DeskProto/8.0/Samples/
Linux: ~/.local/share/Delft Spline Systems/DeskProto/8.0/Samples/

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 where DeskProto will load all driver files (machines, postprocessors and cutters). Any valid new driver files that you copy to this folder will be automatically available after (re)starting DeskProto. Changing this location will make DeskProto unload all drivers (it will 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 Drivers location for DeskProto is:
Windows: C:\ProgramData\DeskProto 8.0\Drivers\
MacOS: ~/Library/Application Support/Delft Spline Systems/DeskProto/8.0/Drivers/
Linux: ~/.local/share/Delft Spline Systems/DeskProto/8.0/Drivers/

In Windows the default location for this type of files is in the C:\ProgramData\ folder (as folders in \Program Files\ can be accessed only by users with administrative privileges). For some reason Microsoft has made ProgramData a hidden folder: to make it visible in File Explorer ('My Computer”) select tab page View and check the option to show "Hidden items".
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 may be a bit confusing it might 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.

DeskProto is available all 'major' languages. Use button **Available languages** to select the language that you want to use for the user-interface. The default language is English, though the Setup may have selected a different language on your request.
For every language a sub-folder in the DeskProto installation folder is present, containing all translation files for that language. The name of the sub-folder is the two-letter codes for that language (conform ISO 639).

DeskProto currently supports the following languages:

de	Deutsch	German
en	English	English (this is the default, no qm files needed)
es	Español	Spanish
fr	Français	French
it	Italiano	Italian
nl	Nederlands	Dutch
pl	Polski	Polish
ru	Русский	Russian - no longer supported, due to the trade boycott because of the war in Ukraine. Sorry!
zh	中文	Chinese, simplified
tw	繁體中文	Chinese, traditional
ja	日本語	Japanese

For each language "nn" the following file is **required** to be present in that folder: *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 8.0\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*

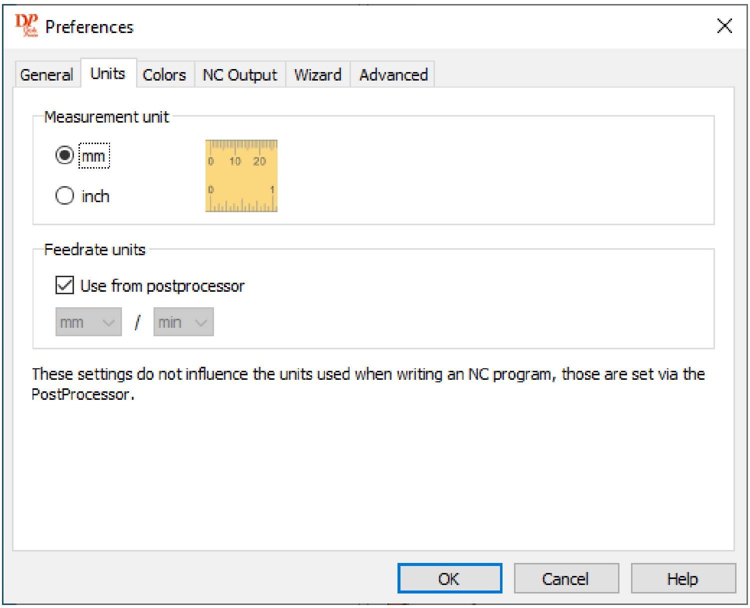
- translations for the custom wizards, in file *DeskProto_Wizards_nl.qm*
If such optional file is not present (or incomplete) the missing texts will be displayed in English.

The cutter files that are installed with DeskProto V8 already contain proper names (translations) for all these languages.

The translated Setup will also install translated versions of the license agreement and the Readme file:
License_nl.rtf and *Readme_nl.rtf*

The last Preference on this page is about **Updates**: the option **Check for updates when exiting the program** will make DeskProto automatically perform the Update check that is available in the Help menu. As this automatic check is invisible when no updates are present (and also is very quick) you will not notice that it is done. Unless of course an update is present. Most of these updates are free, so we advise to keep this option checked.

Units tab page



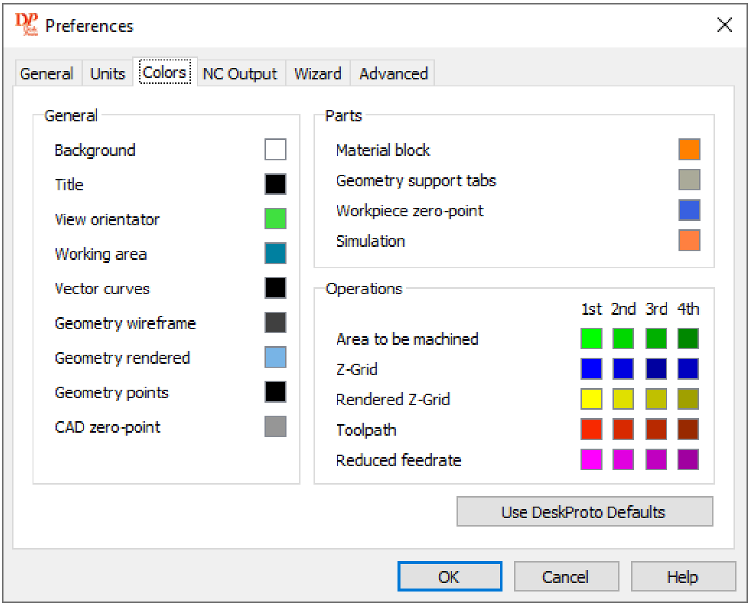
With the option **Measurement unit** you can choose between metric (mm) and imperial (inches) for your DeskProto configuration. This setting will be applied in the user-interface and when loading and saving CAD-data: most CAD files do not state the units used and DeskProto has to assume that they are in the same unit as the one you define here.

The **Feedrate units** to be used in the NC file are set in the postprocessor, the option **Use from postprocessor** lets DeskProto use these units in the user interface as well.
In case you prefer different units for your user interface you uncheck that option and set the units that you want to be used in the user interface. DeskProto will then convert the feedrate that you enter (in user-interface units) to the correct value in NC file units.

This double option to set the feedrate units is handy for instance for inch-addicted users who have a machine that understands NC files in mm only.

Settings that you can check in case of unit problems:
This setting (Measurement unit), in Options> Preferences> Units 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: in the G-code use G21 or G71 to set the machine to use mm.
If working in inch: in the G-code use G20 or G70 to set the machine to use inches.

Colors tab page

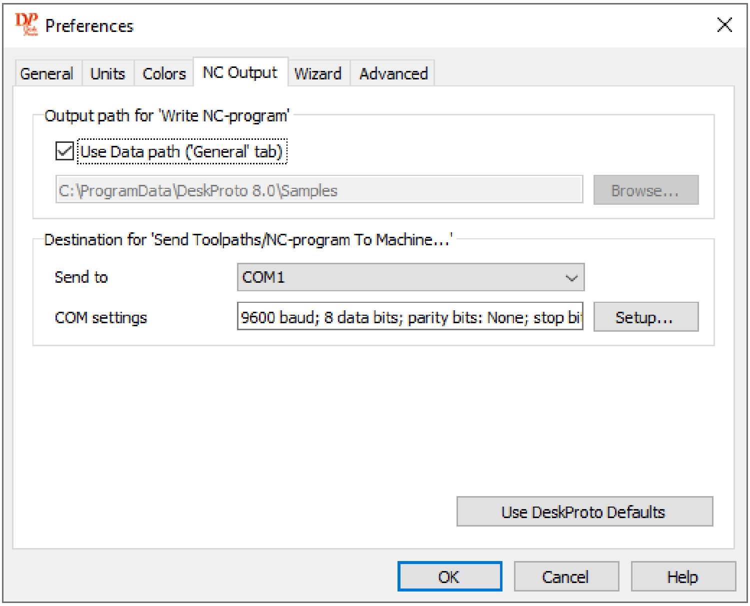


The tab page Colors makes it possible to customize the colors used in the DeskProto view window. Each color fields is a button that you can click on, which will open the Select Color dialog.
As you can see: different operations can be assigned different colors. In case of more than 4 operations: the colors for operation No. 1 will also be used for No. 5, 9, 13 etc.

The option Use fog (that was present here in earlier versions of DeskProto) has been moved to the Items Visible dialog.

In case you have made a gigantic 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: they are about exporting the toolpaths to NC files or to other destinations.

With the first preference you can fine-tune the **Output path for "Write NC-Program"**. 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 later be plugged into the machine).

The second preference, **Destination**, is meant for the commands Send Toolpaths to Machine in the Create menu and Send NC-program File to Machine in menu Create > Extra. By default these two commands are disabled, here you can enable them by determining which process has to be started after giving the command.
This feature is not available in MacOS and in Linux: only in Windows. Sorry about that.

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 toolpaths to Machine and Send NC-program To Machine will not be available (grayed out), and the toolbar 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 simply 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 for ‘Baud Rate’, ‘Data Bits’, ‘Parity’, ‘Stop Bits’ & ‘Flow Control’).
Note that this direct output will only work for some machines (that have an intelligent controller). For most small machines that are connected to the PC using a hardware port it is needed to use **control software** to send the toolpaths to the machine.

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 to **not** 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.

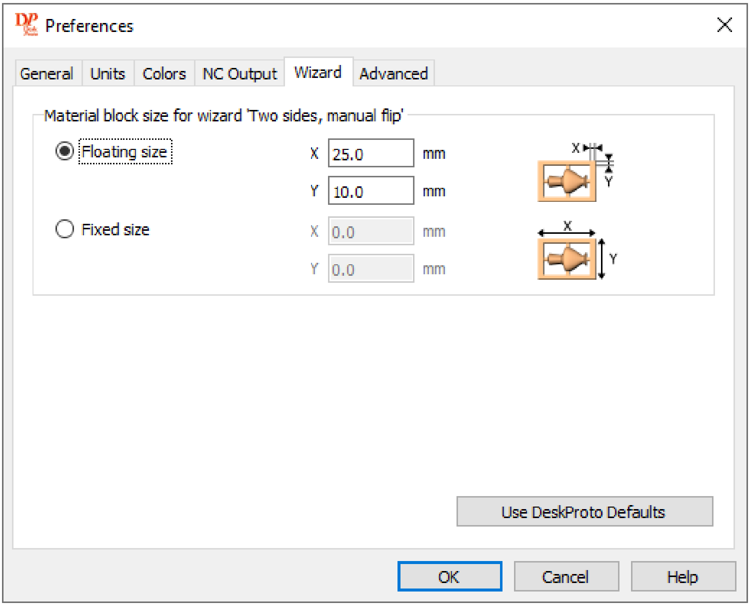
The idea is to configure the **control software** of your CNC milling machine here. This works for many programs, for instance:
- Kay (configure "*C:\Galaad\KAY.EXE*", or "*C:\Program Files (x86)\Galaad\KAY.EXE*").
- PlanetCNC (configure "*C:\Program Files\PlanetCNC\PlanetCNC64.exe*", or PlanetCNC32.exe 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:
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 8.0*) and that image will be shown on the button. Scaled down to the proper size for a toolbar icon.

Button **Use DeskProto defaults** will restore the default settings of DeskProto (Use data path, Documents folder, Send to "None").

Wizard tab page



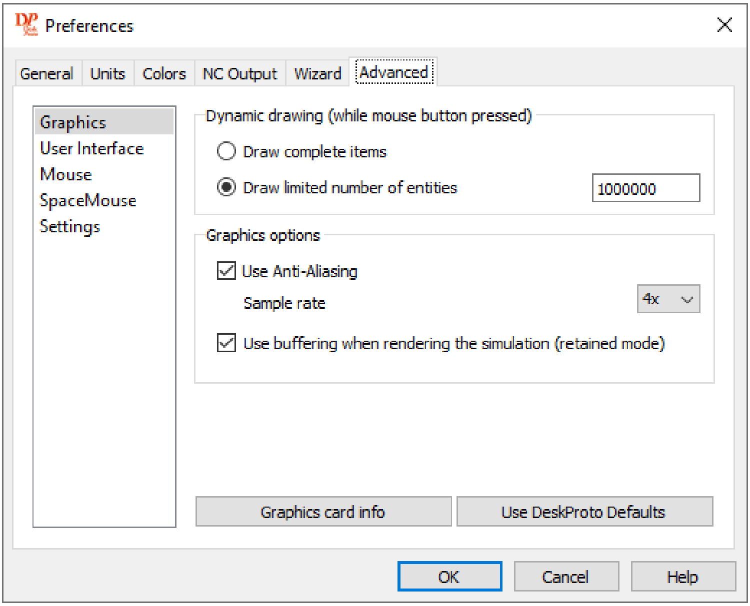
The tab page Wizard contains a preference that you can set for the Two-Sided Milling Wizard (wizard "*Advanced geometry: Two sides, manual flip*"): the size of the material block for two-sided machining. This tab page is not available in the Free edition and the Entry Edition. Two options are available:

- **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 a frame is added. In the edit boxes for X and Y you can enter how much needs to be added as frame. 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.

Button **Use DeskProto defaults** will restore the default settings of DeskProto (floating size, X 25, Y 10).

Advanced tab page



This tab page of the Preferences contains four or five different sub-pages, which you can select by making it active (blue or grey background) in the list on the left. Four sub-pages are always shown: **Graphics**, **User Interface**, **Mouse** and **Settings**; page **SpaceMouse** is available only when DeskProto finds a SpaceMouse attached to the PC. Help for all five 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 of entities 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. More OpenGL options may be available in the driver software of your graphics card.

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 therefore 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 (by Open GL) 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 improvement in quality 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 when rendering the simulation** 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, which may make the drawing process slow.

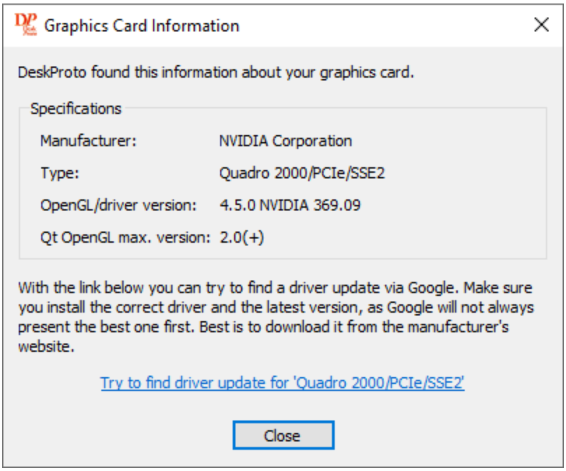
Using Buffering means that all triangles are sent to the 3D **video memory** of the graphics card as 3D data (a 'buffer'). Changing the view then will be completely handled by the graphics card: DeskProto only needs to send the new camera position, the graphics card will do the rest. This is what DeskProto does when displaying geometry data, and for a simulation the default status of this option is to have it checked. On most current PCs that works fine.

The drawback of buffering is that it requires more graphics memory (video RAM). 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 method even slower than the un-buffered method. So when drawing a complex simulation (on the screen of your old PC) is very slow you can try if is helps to uncheck this buffering option.

Without buffering the color values for all triangles for the current view settings are generated by DeskProto, and then 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.

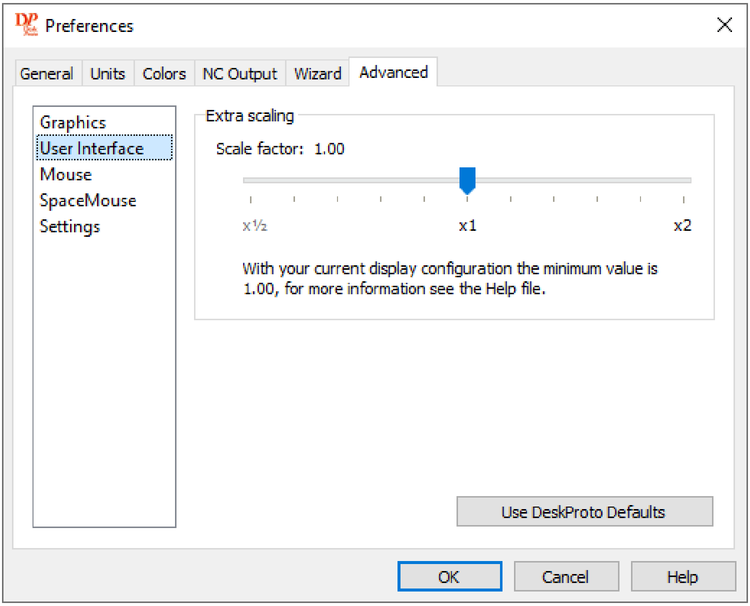
One hidden graphics option is available as well: the **line width** (default 1 pixel) can be changed for all lines in the drawing (curves, blocks, toolpaths). This has been implemented as "hidden feature". Setting DrawingLineWidth may have a value from 1 (default) to 99. Changing this value needs to be done before DeskProto is started.

You can access this setting as follows:
For Windows change registry key HKCU\Software\Delft Spline Systems\DeskProto8.0\Preferences\Advanced\Graphics\DrawingLineWidth
For MacOS change line "8·0.Preferences.Advanced.Graphics.DrawingLineWidth" in file DeskProto.plist
For Linux change line "Preferences\Advanced\Graphics\DrawingLineWidth" in section "[8.0]" of file DeskProto.conf.
Only edit the registry or these files when you are qualified and know exactly what you are doing !



Button **Graphics card info** will show the above screen, with information about the graphics card on your PC..

Button **Use DeskProto defaults** will restore the default settings of DeskProto (draw one million entities, use anti-aliasing rate 4, use buffering).



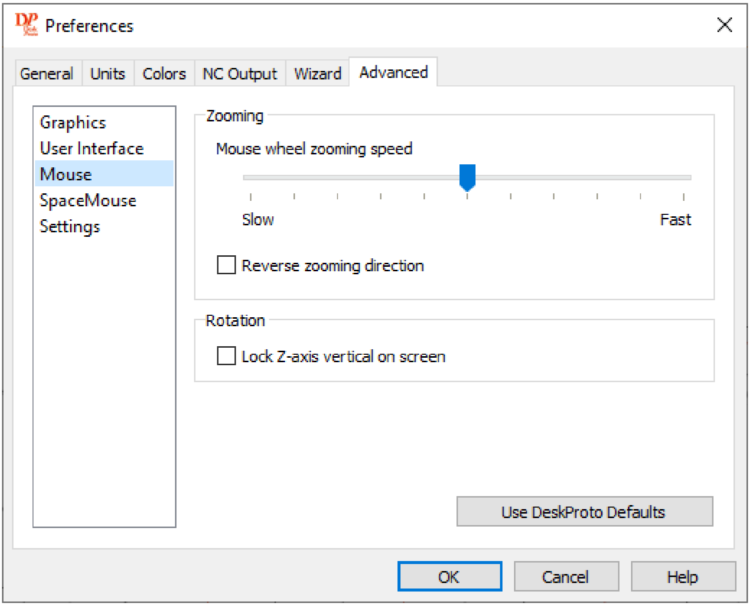
The picture above shows the **User Interface** sub-page of the Advanced Preferences.
The user interface displays both text and graphics, this setting concerns the size of these items. The toolbar buttons for instance are drawn as 32x32 pixel images (for the default toolbar button size 'Medium'), all texts and all images in the dialogs have a fixed sizes as well. At least, on a 'standard' display.

On **high-res displays** the fixed size as just mentioned will be too small for an easy readability. Such displays are called "*4K displays*" (having a horizontal resolution of around 4,000 pixels) or "*UHD displays*" (Ultra High Resolution). When using a desktop size monitor the text and the buttons will be unreadable when the standard number of pixels is used (scaling 100%).
The trick is to set a higher scaling factor in your operating system: say 150% or 200% (for Windows this can be done in the Display settings). DeskProto then will automatically make sure that both the text and the images will remain sharp by giving them a higher resolution.

In case that does not solve the problem this option **Extra scaling** can be used to adjust this automatic scaling factor: the automatic factor will be multiplied by the extra factor. So if the automatic scaling is factor is 2, you can for

instance set the extra factor on 1.5 to configure the total scaling factor to be 3. This will help in case you have trouble reading from the screen. Or when you feel that all user interface items are too large you can set the extra factor on 0.5 to configure a total scaling factor of 1.
The total scaling factor may not be lower than 1, so on a standard screen (96 DPI) an extra factor below 1.0 is not possible.
You may have to experiment a bit to find which settings are best for you - on your monitor.

The button **Use DeskProto Defaults** resets the extra scaling factor to 1.



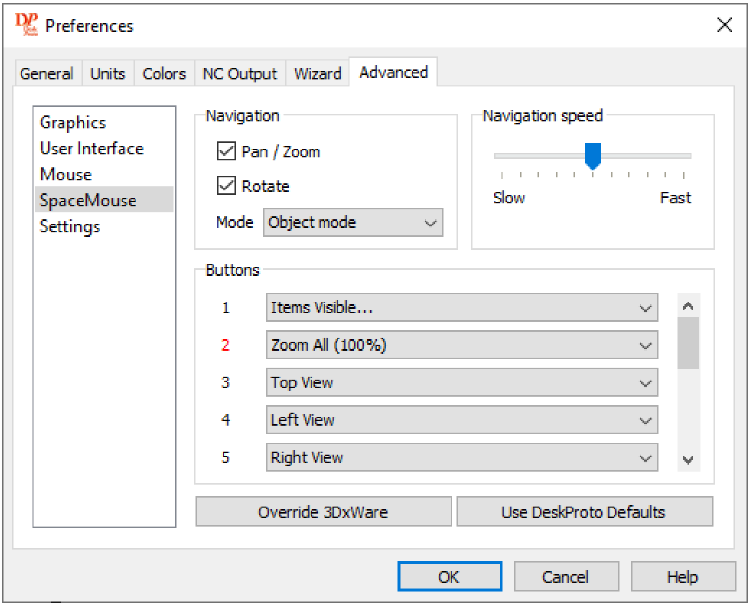
The picture above shows the **Mouse** sub-page of the Advanced Preferences.

Here you can configure two Mouse settings: **Zooming** and **Rotation**.

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. Some CAD programs handle zooming exactly reversed, and it would be quite confusing when you combine DeskProto with such program. So we have added the option **Reverse zooming direction** in order to keep you comfortable.

The latest option (since DeskProto V8) is **Lock Z-axis vertical on screen**. When you check that option the horizon of your scene will always remain horizontal on your screen (and the Z-axis will remain vertical), no matter how you rotate your view. So this option limits the rotation capabilities. This option has been added as that is the way some CAD programs (for instance Rhino) handle the view rotation. When you are used to that method it is best that all programs that you use rotate that way, this checkbox makes this possible for DeskProto.



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 in fact no settings seem to be required. Still DeskProto offers a few.

Navigation 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 will work - which does not seem to make sense though.

The **Mode** that you set determines if the SpaceMouse actions will be applied on the part or on the camera. Two options:
Object mode - SpaceMouse actions are applied on the part, so when you pan to the left the part will move to the left.
Target camera mode - SpaceMouse actions are applied on the camera, so when you pan to the left the part will move to the right. A rotation to the right also means a part rotation to the left - so it does not mean that you rotate the camera.

Navigation 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. If you do not like the default speed you will have to experiment a bit to find the speed that you like.

Buttons:

Each SpaceMouse features (at least) two buttons: for the basic model the left button is button 1 and the right button is button 2. 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.
If nothing is visible in this box: press button "Use DeskProto defaults" to fill it.

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 -> Back 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. The pull-down list contains 22 actions that you can choose from. In this list of actions the words Tilt, Spin and Roll are used: see the image above for an explanation. 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.

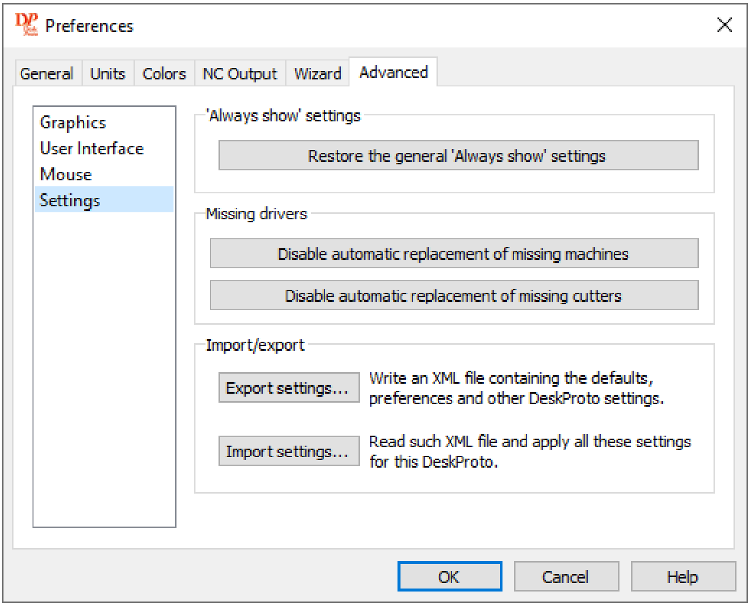
Button **Override 3DxWare** is visible only in the Windows version of DeskProto.
To understand the function of button Override 3DxWare, it is important to know is that DeskProto does **not** use 3Dconnextion's software development kit (**3DxWare**), and also does **not** use the 3DxWare driver that 3Dconnexion 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 8.0\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. The Always show option is available in two versions: general and project specific. In the latter case it is valid only for the current project, and will be stored in the project file.

What to do when you want the warning restored ?
Using button **Restore the 'always show' settings** you can restore all general warnings (so it will again set the mark in all these checkboxes).
The project specific warnings can be restored in the Project Parameters.

Missing drivers
When you open an existing project on a different computer it may happen that the project needs a machine definition that is not present on that new computer. DeskProto makes it possible to automatically replace the missing machine and/or missing cutter(s) by a definition that is present on the new computer. For instance a definition for the same cutter, however with a slightly different name.

These replacement options are described on pages Missing machine and Missing cutter.

The two buttons to **Disable automatic replacement...** do exactly what this text tells you. When you press a button DeskProto will then also forget all "*unknown - replacement*" combinations that you have set for this driver type.

Import/export
All DeskProto defaults and preferences are stored on your computer.
For Windows this is done in the Registry, conform the specifications for Windows applications as made by Microsoft.
For MacOS the defaults and the preferences are stored in file *~/Library/Preferences/com.delft-spline-systems.DeskProto.plist*
For Linux the defaults and the preferences are stored in file *~/.config/Delft Spline Systems/DeskProto.conf*
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 tools allow you to export DeskProto's 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 use of these commands is straightforward: button Export settings will export an xml file, button Import settings will import one. The file format used is an XML file, default filename for export is *deskproto_8-0_settings.xml*

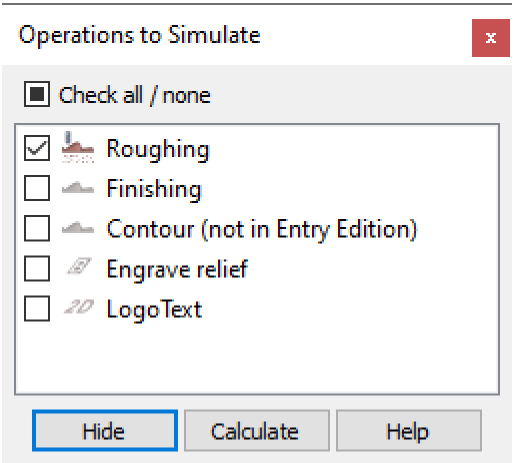
The resulting V8.0 XML files can be used for all three operating systems (Linux, MacOS, Windows). In DeskProto V7.0 the Windows version used a different format for this XML file. DeskProto V8.0 can import that old format, however DeskProto 7.0 cannot import an XML file that was written by DeskProto V8.0. DeskProto V7.1 and V8.0 use the same format (though of course not all settings are present for both versions).

The export option will write the current settings, so for the current version of DeskProto.
The import option of DeskProto V8.0 accepts XML files, written by V6.1, V7.0, V7.1 and V8.0. DeskProto V6.0 and older versions will write a .REG file when exporting the settings: DeskProto V8.0 cannot read such .REG file.

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\

Note 2:
Importing and exporting only concerns the settings (that are stored in the Registry). The **driver files** (machine, postprocessor and cutter definitions) are not included.

4.21 Simulated Operations



This dialog is visible when a Simulation is visible on screen
Removing this dialog (red cross icon or button Hide) 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). Pressing again will remove the simulation.
- using the Show 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.

A simulation is made for one **complete part**. This Simulated Operations dialog will pop up in which you can select which operations should be included in the simulation. When no operations are yet processed 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 black 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 or switching the simulation off in the Items visible dialog will all make the simulation disappear from the view.

Note that the simulation calculation algorithms for 3-axis toolpaths and for rotation axis toolpaths are completely different.

- The XYZ algorithm (**3-axis toolpaths**) uses a grid on the XY plane with a Z-height for each grid-cell: the **grid-based simulation**.
- For the XZA simulation (**rotation axis toolpaths**) this is not possible, this calculation is based on voxels (small "3D pixels"): small cubes that either are filled with material or not: the **voxel-based algorithm**.

The voxel-based algorithm is much more complicated, and so calculation time will be higher. The quality of the resulting image on the other hand is higher for the grid-based simulation (as for voxel-based the small cubes are visible).
One more difference is that the voxel-based simulation removes material only where the cutter can actually cut, defined by its cutting length. For a finishing operation this might be completely within the block: creating an invisible cavity in the block, So for a voxel-based simulation it is important to simulate the operations in the correct sequence: first roughing, then finishing.

4.22 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).

Shortcut:

Keys: Ctrl+B (Windows, Linux) ⌘+B (MacOS).
B stands for "Begin", as the S already is being used (for Save project).

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:

- Start a new project
- Open an existing project

- Other tasks

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

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.
When starting a new project using dialogs you need to select which project type. The difference between Vector/Geometry/Bitmap projects is explained on page New project.

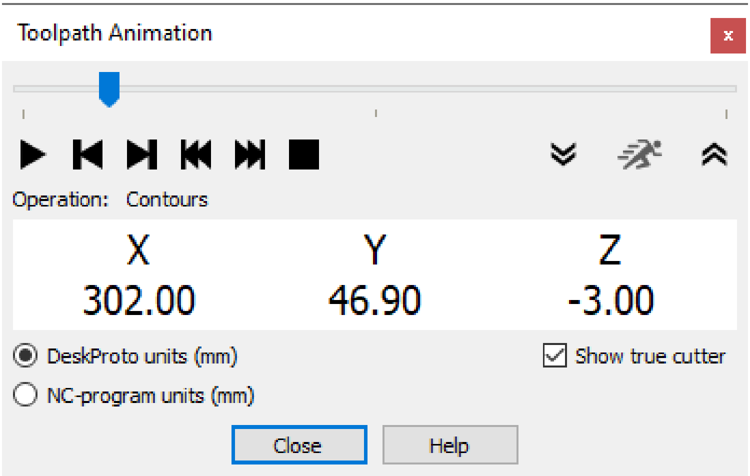
The DeskProto Setup has installed a number of **Sample projects** and **Sample geometries** on your PC. Including some great geometries like the DeskProto picture frame: see the lessons in the DeskProto Tutorial book.
Conform Microsoft's specifications the **Samples** have been installed in the \ProgramData\ 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 (no administrative rights) other locations are not permitted by Microsoft. Also when using MacOS or Linux the Samples folder may not be easy to find for you.
Using the checkbox **Use samples folder** will make DeskProto open the sample folder when you need to load the CAD data.

To open an **existing project** two options are present:
Command **Open project** will allow you to open any project file by browsing one.
Recent projects will be available only when you have used DeskProto V8 before. In this dialog only the latest 5 projects will be shown, more may be visible when you select this option via the File menu.

In the **Other tasks** section, **Open sample project** makes it easy to find the sample projects.
The other three 'other tasks' (Read tutorial book, View tutorial videos and Visit user forum) 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.

4.23 Toolpath Animation



A DeskProto screen with many toolpaths drawn may be difficult to interpret, being one big *'mess of red lines'*. The Toolpath animation option offers help, by making the cutter draw the toolpath while moving over your screen: an animated view of the toolpaths.
The animation concerns one complete part, however, only the toolpaths of the visible operations are drawn, if needed they will be calculated first. In case more than one operation is present you will see small vertical lines below the slider bar, indicating the start of a next operation. The default speed of the animation will be as defined by the feedrate that you have set,
You can reach this option via the Create menu ("Show animation") or via toolbar button 

The dialog shows the following icon buttons:

 The **Slider** bar allows you to quickly move to any location in the toolpath. Simply click on the blue knob and move that to a different position on the horizontal bar.

 The **Play** button will start the animation.

 Button **Previous** will move the cutter to the previous point on the toolpath. These points can be shown by checking 'Show points' for the toolpaths, in the Items visible dialog.

 Button **Next** will move the cutter to the next point on the toolpath.


Button **Begin** will move the cutter to the start of the toolpath for the current operation, or (if already at the begin) to the start of the previous operation.


Button **End** will move the cutter to the end of the toolpath for the current operation, or (if already at the end) to the end of the next operation.


Button **Stop** will halt the animation.


This icon is not a button. It shows that the buttons on either side are to set the **Speed** of the animation.


Button **Slower** will reduce the speed of the animation.
The Speed icon will be replaced to show the current speed setting, for instance "1/2x"


Button **Faster** will increase the speed of the animation.
The Speed icon will be replaced to show the current speed setting, for instance "4x"

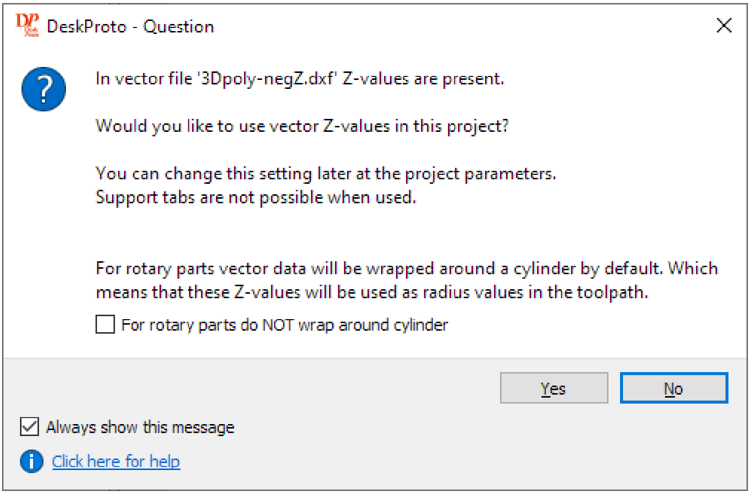
Below these icons DeskProto will display the name of the **Operation** that has created the toolpath at the current cutter position.

The coordinate values that are shown are of the points in the toolpath, so the points that are present in the NC file. While the cutter is moving the dialog shows the coordinates of the point that the cutter is moving to.
The image above shows coordinates for X, Y, and Z, for a rotary toolpath the dialog will show X, A and Z.

The units that are used to display the coordinates can be either the **DeskProto units**, as set in the DeskProto Preferences or the **NC-program units**, as set in the Postprocessor. For some machines the latter values may be quite different, for instance for machines that require coordinates in 1/1000 of a mm.

The option **Show true cutter** does what it says: the cutter will be drawn with its true shape and on scale. When this option is not checked the a symbolic 'icon' is drawn to represent the real cutter. Drawing the 'true' cutter is optional as it may obscure the toolpaths and may make the animation slower.
When you are working with a Special cutter that is very large (compared to the part) it can happen, when the 'True cutter' is shown, that the cutter is not drawn completely as it partially falls outside the 3D Viewport (the 3D space that is rendered).

4.24 Vector data - Use Z-values

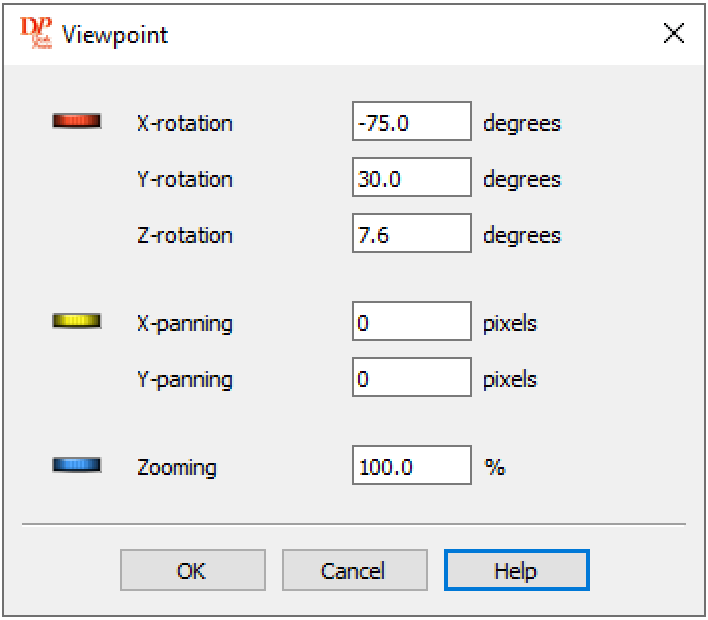


For Vector data two different data types are possible:
2D Vector data - no Z-values are present at all or all Z-values equal to 0.0
3D Vector data - Z-values are present that are not equal to 0.0.

The vector file that you now opened is from the second type, so it contains vector curves in 3D.
You now can select whether or not you want to use these Z-values: if you say No all Z-values in the vector curves will be reset to 0, so the data will be converted to 2D vector data.

When you want to use the Z-values (answer "Yes") you can enter a second setting, about how to uses this file for rotation axis machining. This is the checkbox "**For rotary parts do NOT wrap around cylinder**".
The difference between wrapping and not wrapping is explained on page Rotation axis machining.

4.25 Viewpoint



The View menu command Viewpoint -> Custom... displays the **Viewpoint dialog**, in which you can change the settings of the active view, which are the settings for rotation, panning and zooming. In fact this is setting the Camera position.

Shortcuts:

Toolbar: 

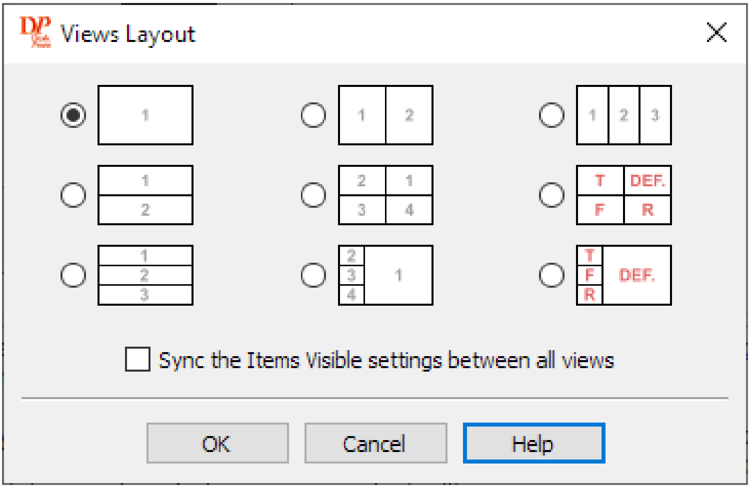
Mouse: Right-click in a view, and select the Viewpoint item in the context-menu.

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 for rotation, pan and zoom exactly, using the keyboard. The rotations are executed in the order X, Y, Z. The effect is the same as when using mouse rotation, mouse pan, mouse 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.

4.26 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 for 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.

When the checkbox **Sync the Items Visible settings between all views** is checked all views show the same items. So when you add an item to one view it will be added for all views. When checking this for the first time the settings of the **active view** (marked by a blue border) will be copied to the other views.

Shortcuts:

Toolbar: 

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.

V Concepts

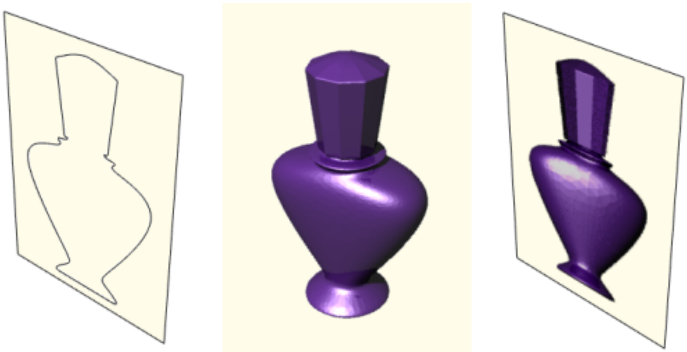
5.1 CAD Data

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 in DeskProto in all cases is loading some CAD-data, as DeskProto does **not** offer **CAD** (**C**omputer **A**ided **D**esign) functionality. You will need to use other software to design your parts.
DeskProto supports three different types of **CAD-data**:

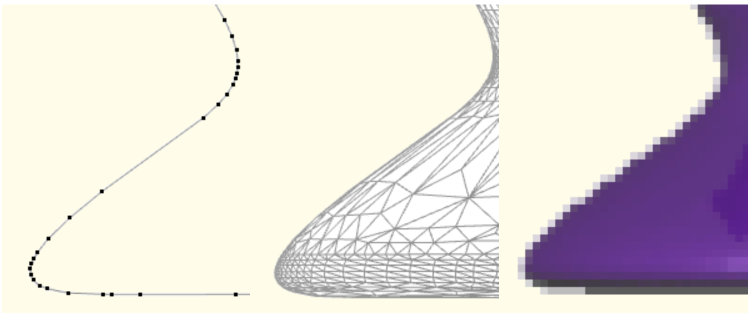
- Vector data One or more curves: points in space connected with straight line segments (or with arcs).
- Geometry data A collection of small triangles that describe the outer surface of a 3D shape.
- Bitmap data A grid of colored pixels on a flat plane, forming a 2D image.

The images above show all three types. For Geometry data and Bitmap data the images are identical, because a 2D screen or page we can only disply bitmap images: also for 3D CAD data. Bitmap data for DeskProto normally will be a digital photo, a company logo or similar. The next image shows 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 (though DeskProto will also accept 3D curves as Vector data).



Vector data **Geometry data** **Bitmap data**

The images above (zoomed in on the base of the bottle) 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 powerful feature in DeskProto is that it can combine these types of CAD data in one project: combine 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.

5.1.1 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 and SVG are supported):

- File menu command Save Vector data as...
- Button Save as... on the Vector tab of the Project parameters

The first option saves all Vector data to one file (also when various Vector files had been loaded), the second only that data of the selected vector file.
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 Data Options dialog.

Vector curves can be either **open** curves, **closed** curves or single **points**. This is important as Pocketing and V-Carving are possible only for closed curves.
When reading vector curves DeskProto will check whether or not a curve is closed: it is closed when the end-point is identical to the start-point. Or in fact almost identical, as DeskProto applies a tolerance: when the difference is less than 0.001 mm the points are considered to be equal. The same tolerance is used when separate open curves can be combined to one long curve (also see the option 'Preserve curve direction' in the Project parameters.
When a curve that seems closed is in fact open you can find the error by checking "Show points" for Vector curves in the Items Visible dialog. The start point of the curve will be indicated by a larger dot. Closed curves do not have a starting point, so a large dot indicates a gap.

The vector calculation algorithm that DeskProto uses only works for polylines. So when opening a vector file that contains arcs, circles, ellipses, splines or other curves each of these curves will be converted to a polyline. The size of each straight line-segment in the polyline depends on the radius of the arc. If you are interested, the exact formula for arcs is:

```
stepsize = 2 * acos( (radius-0.001)/ radius )
```

As a nice extra feature DeskProto also supports **3D Vector files**. Most vector files do not contain any Z-coordinates: you will then 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-values defined in the file.
When you open a DXF file containing Z-coordinates, 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 "Use Z-values" 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: The vector curves then will be placed on the zero level of the geometry (in CAD coordinates). When the option is not checked the curves will be placed on the top of the block.

When **rotation axis machining** two workflows are possible for 3D vector files:
- Wrap around cylinder (as with 2D files)
- Do not wrap, see the page about rotation axis machining. This setting as well can be found on the Vector tab of the Project parameters.

Text in a vector drawing that is stored as a series of **ASCII character with a font definition** cannot be read by DeskProto: each character needs to be stored as curves.

Vector File types

For Vector data DeskProto supports four file types:

- **DXF** the most widely used exchange format for **engineering**
- **SVG** open source format meant for internet, supports vector, bitmap, interactivity and animation.
- **AI** widely used format by Adobe, for **graphics design** (only version Illustrator 8)
- **EPS** graphics file format by Adobe (only version Illustrator 8)

For all four file-types DeskProto supports (so can read) only a small subset of all entities that can be present in the file, see the lists below.

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. DeskProto will convert the arc to a polyline (many small line segments), as the algorithm does not 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.

Internally DeskProto only uses polylines for vector data, so all entities will be converted to polylines (see the information above). When saving vector data in a DXF file it will thus contain only polylines.

Note that DeskProto also supports Geometry data in DXF files.

SVG

The acronym stands for **S**calable **V**ector **G**raphics, an open standard developed by the World Wide Web Consortium (W3C).
It can contain many different entities, DeskProto supports a small subset only.

Supported SVG entities for Vector data in DeskProto:
line
rect
circle
ellipse
polyline
polygon
elliptical_arc
cubic_bezier
quadratic_bezier

Internally DeskProto only uses polylines for vector data, so all entities will be converted to polylines (see the information above, for DXF). When saving vector data in an SVG file it will thus contain only polylines.

When the size in SVG file is only in pixels (no real life dimensions in the file) DeskProto assumes 4 pixels per mm (101.6 dpi), as that comes close to most screen resolutions.
In the SVG format the zero point is in the upper left corner of the screen, with the positive Y direction going downwards. In DeskProto (and on the machine) the positive Y direction goes upwards, so DeskProto needs to mirror all Y values.

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 a 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.
Adobe changes the file formats for AI and of EPS with every new version of Adobe Illustrator, which makes these files not really suited for data exchange. The most widely used version is the one used in **Adobe Illustrator 8**, and that is the version that DeskProto supports. **So AI files need to be saved/exported for Adobe Illustrator 8 to make them readable for DeskProto**. Loading an AI file or EPS file in a newer version 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. Here you can also find the option to show all points in each vector curve.

5.1.2 Geometry Data

DeskProto can load three types of **CAD data**: Vector data, **Geometry data** and Bitmap data.

For Geometries DeskProto only recognizes "Polygon data": the outer surface of the geometry described using a (large) number of triangles. This type of geometry is also called mesh data, and any 3D shape can be defined this way. 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.
DeskProto will generate the toolpaths by projecting a flat toolpath pattern over the 3D geometry.

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 Geometry 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

The first option save all Geometry data to one file (also when various geometry files had been loaded), the second only that data of the selected geometry file.
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 Data Options dialog.

Geometry File types
DeskProto supports four geometry file types: **STL**, **3MF**, **DXF** and **VRML**. The only type that is fully supported is STL, as that file can only contain triangles, nothing more. 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. We do not yet have much feedback about working with 3MF files.

Two other file types that are often 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 error possible

STL

This file format has been 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 a 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 a small gap (say a 0.001 mm gap between two triangles) still is 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. The direction of each normal should point outwards. When the option 'Skip Backfaces' in the Project parameters has been checked the backsides of the triangles will not be considered: drawn in red, 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: in an STL file the vertices are listed in counterclockwise order when looking at the object from the outside.

3MF

The **3**D **M**anufacturing **F**ormat has been published by the 3MF Consortium, as an open source file format for additive manufacturing. Just as STL it defines the geometry as polygon data, but it includes more information about the part than STL offers, for instance about material and color. DeskProto does not use this extra information: only the information in group "3D Model" is read (the geometry information).

3MF files will be smaller than STL files at 3MF is a compressed format (ZIP file).

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 when loading geometry data.

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").
It has become an obsolete filetype, only very rarely used.

Corrupt geometries
In the past not all 3D CAD systems exported valid geometry files; loading an invalid file may result in geometries that do not look right on the DeskProto screen. With an up-to-date CAD program this is not likely to happen though: 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 red. 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 red. 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 simply accept files containing cracks, holes, orphan surfaces and many more inconsistencies. Files that would be rejected by most 3D Printing programs.

Viewing the Geometry data
After loading, you can see the Geometry data on your graphics screen. As rendered geometry, as wireframe, as wireframe with hidden lines removed, or as points. To determine how to view the geometry you can check or uncheck the items in the Items visible dialog.

5.1.3 Bitmap Data

DeskProto can load three types of **CAD data**: Vector data, Geometry data and **Bitmap data**.

A Bitmap file contains a 2D image 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 for each pixel, in a bitmap complex pictures are possible. Digital photos for instance are stored in bitmap files.

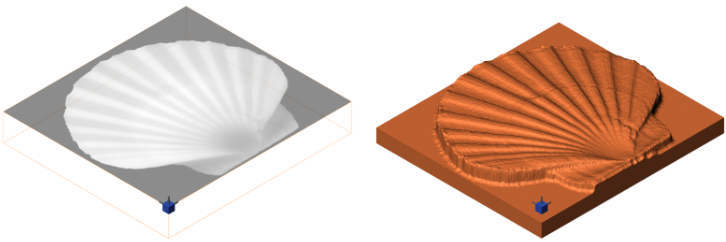
DeskProto needs an image in gray values, it will automatically convert color pictures to Black&White (B&W) when loading. In case you are interested: the conversion formula is *gray-value = 0.30 * red + 0.59 * green + 0.11 * blue*.

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 to a Z-grid, that can be used for toolpath calculations.

The bitmap image will be displayed on the top surface of the block. When combined with other CAD data it will be positioned with its bottom left corner at point X=0 Y=0 (in CAD coordinates), still at the top of the block.

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 right 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 (512x512 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

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 for export, except GIF.

Bitmap File types

DeskProto supports the 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). DeskProto also supports PNG files with a 16-bit grayscale image (65536 shades of gray).
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. DeskProto can read both 8-bit grayscale images (256 shades of gray) and 16-bit grayscale images (65536 shades of gray).
WEBP	**Web P**icture format, developed by Google (as a derivative of WebM, a video format file), intended to make bitmap files very small and thus perfect for website use.

HEIC	**H**igh **E**fficiency **I**mage **C**ontainer (supported only in the MacOS build, and only for import). This is the standard format used by iPhones and iPads to store photos.

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...” (on tab XY Transform that was just mentioned). The edit fields for Custom will show the resulting relief size. Note that when you later change the precision that 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. Per default the bitmap is drawn translucent, making it possible to see items behind the bitmap. In the Items visible dialog that translucency can be switched off.

5.2 Project

A project is the file 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 though 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**. DPJ files are simple text files, that can be edited using a plain text editor.

Project Structure
The information in a project file is structured: in addition to some general information five groups are present:
- General information
- CAD files
- Parts
- Operations
- Views
- NC files (if any)

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.

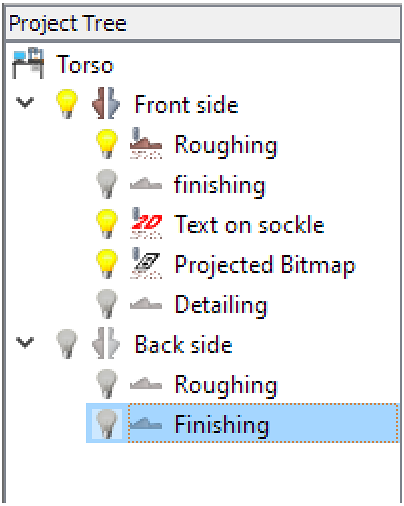
The **Part** is what you are going to machine. A project contains one or more parts (zero parts is not possible), as for one model several parts may be needed. For instance the left halve and the right halve of a hand drill. The part definition consists of a number of parameters, like scaling values (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 maximum number, 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 cutter, 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**.

The project file also contains information about the **View** (camera position) for each part, and about the **NC files** that you have saved when working with this project.

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: the type of operation that is present defines the project type. 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, 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, or by pressing the Open button on the Toolbar. When you open a project the currently opened file will be closed.
The folder that is shown when you first open the Open File dialog is 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, or by pressing the Save button on the Toolbar. 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 useful 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.

5.2.1 Part

A part is what you will machine using one setup (a block being fixtured on the machine). For some products that is all that is needed, for other products more parts may be needed: 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; 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 the first operation is a Vector operation your default project will be a Vector project
- when the first operation is a Geometry operation your default project will be a Geometry project
- when the first operation 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.

5.2.2 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, V-Carving, 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. When more than one operation is present a dialog will open 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 to click on a lamp icon 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 in this dialog.

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; 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.

5.2.3 Geometry Operation

An operation gives a description of how the material should be machined. DeskProto features three different operation types:

- a Vector operation is meant to create toolpaths for Vector data
- this **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. When more than one operation is present a dialog will open 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 to click on a lamp icon 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 in this dialog.

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; 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.

5.2.4 Bitmap Operation

An operation gives a description of how the material should be machined. DeskProto features three different operation types:

- a Vector operation is meant to create toolpaths for Vector data
- a Geometry operation is meant to create toolpaths for Geometry data
- this **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. When more than one operation is present a dialog will open 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 to click on a lamp icon 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 in this dialog.

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; 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.

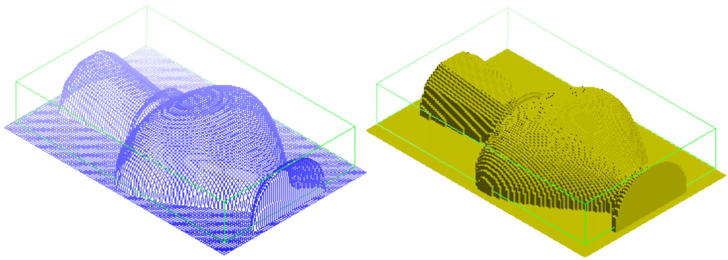
To edit the parameters of the default Bitmap operation, go to the Options menu and select the option Default Bitmap operation Parameters. This will open the Bitmap operation Parameters dialog.

5.3 Process steps

5.3.1 Z-Grid

The Z-grid is an intermediate result, in-between the CAD-data and the toolpaths. 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 is missing in the toolpath, you can 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 the grid cells are too large, or some error is present is the geometry file. Besides that it will 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 for accurate precision settings it will take some time to draw a rendered Z-grid 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. Any undercuts that are present have become solid: for each grid-cell only the highest Z-value is stored. Especially for inverse milling this rendered Z-grid is useful as a preview to show what will be created.
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 that cannot create sharp inner corners).

When creating the Z-grid DeskProto calculates a large number of points on the surface of the geometry, finds the correct grid-cell for each point (based on X and Y) and fills that grid cell with the Z-value of the point. For each cell only the highest grid value will be stored. This explains some characteristics of the DeskProto toolpath: undercuts are not possible, and the resulting part always will be a bit larger than the CAD model. How much larger depends on the Precision values.

When projecting vector curves or a bitmap on the geometry DeskProto will use a Z-grid to determine the correct Z-value for each point in the toolpath.

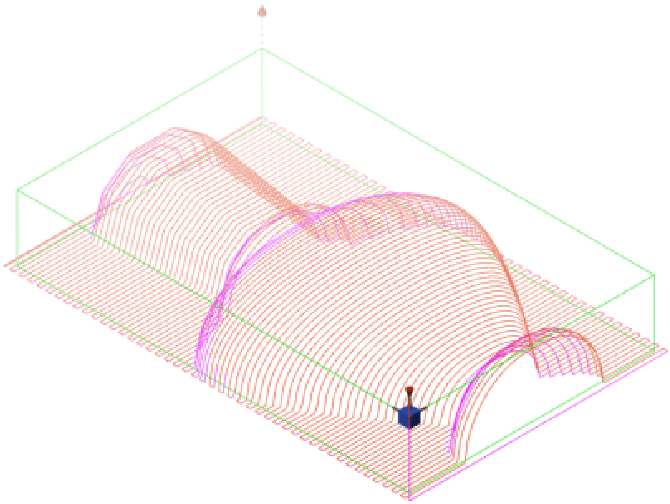
Each operation has it's own Z-grid.

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.

5.3.2 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 image below shows both the start point and the orientator: on slightly different positions.

The toolpath is a very important representation for a visual check before starting the milling machine: any errors should best be found before milling (the toolpath Animation is a good tool to find errors). 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 item will automatically be switched on in the scene (made visible in the Items visible dialog). To send this toolpath to the machine it should first 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.

Options when displaying the toolpaths
As said the toolpaths are drawn as red lines, and rapid movements in dashed gray lines. You can change these colors in the Preferences.
In the Items visible dialog the following three options can be checked:
Show points to add a dot on each point in the path (default is OFF)
Show arrows to show the direction of the toolpath (default is OFF)
Show rapid movements to show or hide all dashed gray lines (default is ON)
One more useful option is present: in an **Animation** you can watch the cutter proceed over the toolpath.

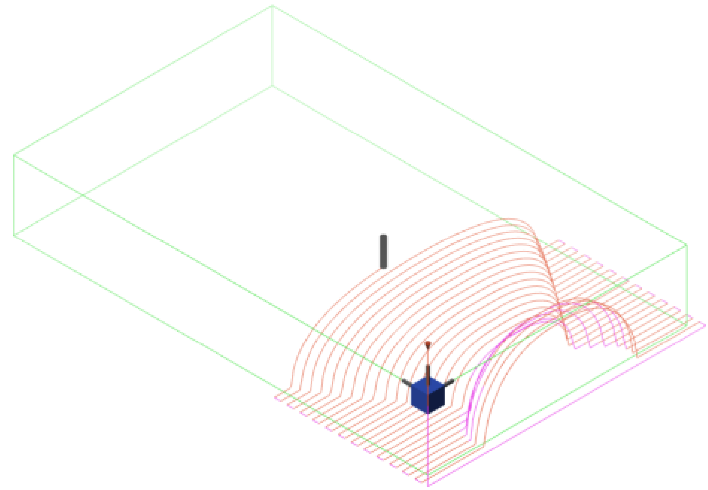
How to calculate toolpaths

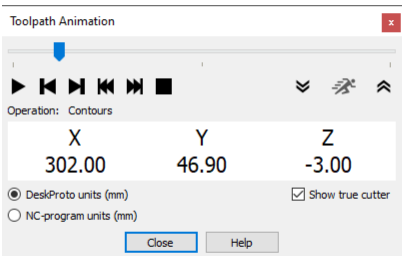
You can let the toolpaths be calculated by selecting the option Calculate Toolpaths under the Create - Extra menu.

As a shortcut this convenient button is present on the Toolbar.
Another way is checking the Toolpaths checkbox in the Items visible dialog.

5.3.3 Animation

Before sending the NC program file to the machine it is always good to check the toolpaths by viewing the drawing on your screen. When many toolpaths are present it may be difficult to interpret the drawing, even when you make most operations invisible. To make it easier to see what exactly happens DeskProto offers the Animation.





How to show an animation

You can show an Animation by selecting the option Calculate Animation in the Create menu.

As a shortcut this convenient button is present on the Toolbar.

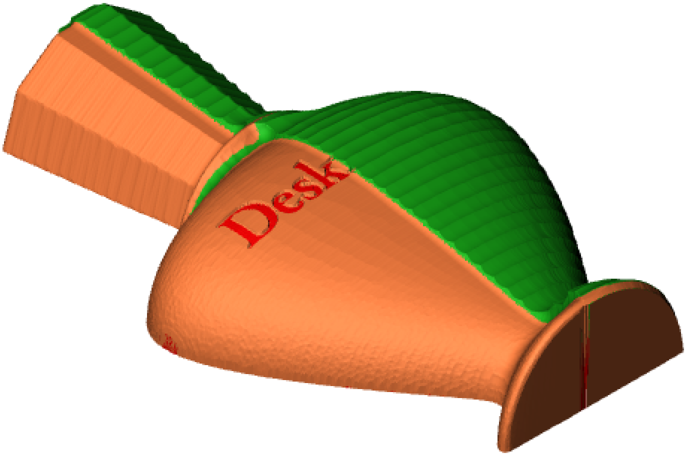
When you open an animation all toolpaths we disappear from your screen, and the Animation dialog (shown above) will pop up. A small cutter will be drawn as well, on the first position of the toolpath. This is a symbolic representation of the real cutter (so size and shape are not correct): for most cutters a small vertical cylinder is drawn in dark gray.

Using the controls in this dialog you can start the cutter movement and make it follow the toolpath. This is a great help, to see in advance how the cutter will move when running this path on your machine.

5.3.4 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 **Simulation**.

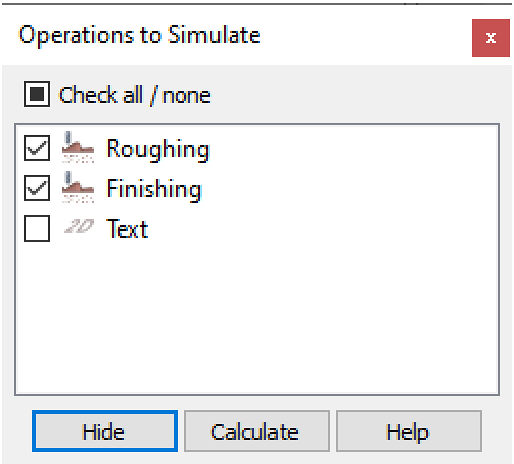
The Simulation is a 3D model, displayed on the screen, that shows you what the resulting part will look like. This can be used to check things like the resulting surface smoothness, error movements that damage the part (if any), rest material where the cutter cannot reach, etc. DeskProto will calculate a simulation in 3D, so you can rotate, pan and zoom it just like any other item on screen. The calculation is done using only the toolpath data, so without use of the underlying CAD data.



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 half the 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 it is not possible to undo the removing of material. Unchecking an operation will restart the simulation with a fresh block of material.
Because of that the Operation's visibility settings in the tree cannot be used here. 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).



DeskProto uses in fact two completely different calculation algorithms: one for 3-axis toolpaths (XYZ) and one for rotation axis toolpaths (XZA):

- the XYZ algorithm uses a **grid-based calculation**: a grid on the XY plane with a Z-height for each grid-cell.
- the XZA simulation uses a **voxel-based algorithm**: a calculation based on voxels (small "3D pixels") that either are filled with material or not.

The voxel-based algorithm is much more complicated, and so calculation time will be higher. The quality of the resulting image is higher for the grid-based simulation, however that algorithm simply does not work for rotary machining.

One more difference is that the voxel-based simulation removes material only where the cutter can actually cut, defined by its cutting length. For a finishing operation this might be completely within the block: creating an invisible cavity in the block, So for a voxel-based simulation it is important to simulate the operations in the correct sequence: first roughing, then finishing.

You can set the **Level of detail** of the simulation on the Simulation tab of the Part parameters. For the grid-based simulations this sets the number of grid-cells to be used (the size of the grid), for the voxel-based simulation the number of voxels.

For Geometry projects the DeskProto grid-based simulation allows you to **compare the simulation with the original geometry**, and indicate any differences with a color (so this option is not available for rotary machining). 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.

5.3.5 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 (or rather to the machine's control software) 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. 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, when Chaining has been applied, even toolpaths of Operations in several Parts can be combined.

NC Files list
DeskProto will 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 (only a few machines support this). 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.

5.4 Types of machining

5.4.1 Two-sided machining

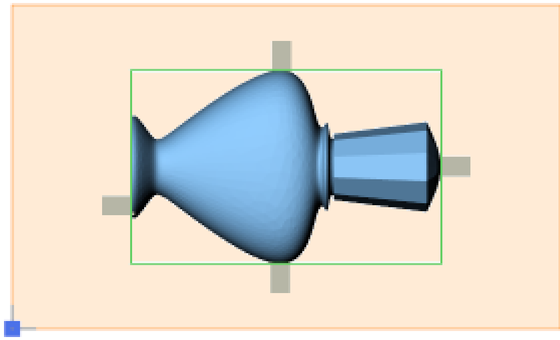
This page is about Geometry machining (so this does not apply to Vector and Bitmap).
Some models can be completely machined from one side, like DeskProto's DpPictureFrame sample geometry. For other models, like the perfume bottle sample geometry, machining from one side is not sufficient.
A complete 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 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.

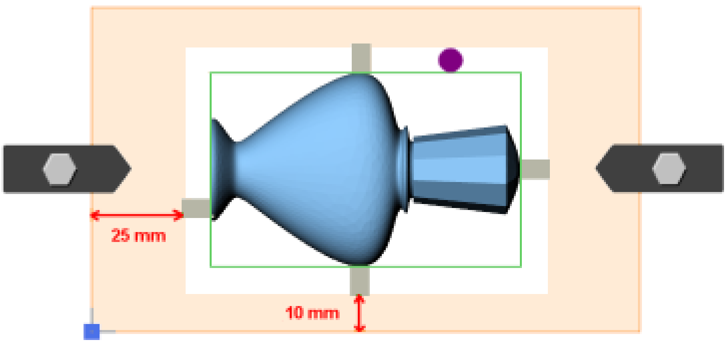
This help page concerns the third method: **Two-Sided machining,** using the manual flip. For the automatic flip see the page on indexed machining.

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 advanced geometry **wizard** "**Two sides, 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 needs to be larger than the part, leaving a frame around the part. This frame permits you (after machining the first side) to again clamp the block in order to machine the second side as well (turned upside down). See the illustrations below, where the orange part is the block (top view).



DeskProto will lock the Operation's Area to be machined to the geometry's bounding box (for X and Y). DeskProto will add a Border on all four sides so the cutter can also machine the outside surfaces of the part. See the image below, in which the cutter has been drawn in the border area.
Obviously the part 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 part to the remaining part of the block (the frame). In the illustration above the four gray rectangles are the Support Tabs.

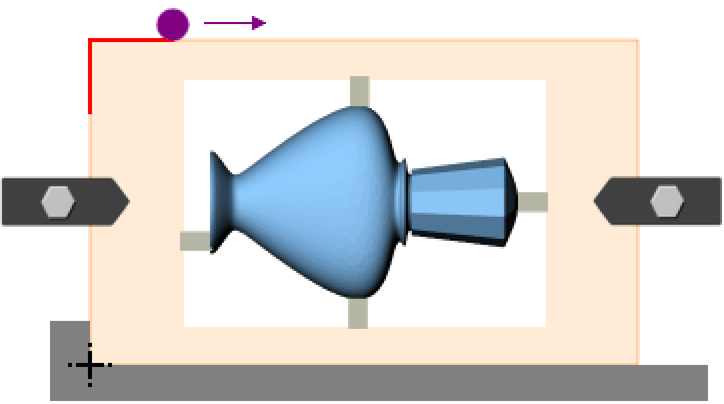


DeskProto will set the width and depth (X and Y) of the material block. For X both left and right 25 mm (or 1 inch) is added outside the support tabs, permitting you to clamp down the block on the machining table at that spot. For Y both front and back 10 mm (or ½ inch) is added, resulting in a block 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 the top surface and the bottom surface are exactly parallel to one another as well.
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 width 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 dimensions to be used 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 that makes the second side exactly fit on the first. For this aim DeskProto uses two rulers (stop-bars) that are exactly parallel to the machine's axes, and that 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 for this small bar: it is in fact an end-stop). 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 red 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 as specified in the **Report file**, and needs to be machined along the complete back. After turning upside-down these two reference planes need to exactly touch the rulers, making sure that the block is exactly lined up with the machine, and exactly at the correct position.
Remember that in order to machine a plane at X=0 the tool needs to move along Y at X=-R mm (or inch), in which -R is "minus the Radius of the cutter". A similar radius correction needs to be used for the second reference plane.

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, combined with matching holes in the block that 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 (this needs to be set after completing this wizard, for both parts).
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 button **Write report file** on the last page of the wizard. It gives all information needed for the actual machining process. DeskProto will automatically open the file in Notepad, making it easy to print the file and/or to save it. Default filename is Wizardlog.txt: save it using a different name to prevent it from being written over by a next wizard.

Of course you are free to 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 Tutorial videos are available as well.

5.4.2 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 (**continuous rotation**)
- the part can be machined from one side (three axis machining), then rotated and machined from a second side, etc. This is called **indexed machining**.
DeskProto supports both methods, obviously only in the Multi-Axis edition of DeskProto.
On some four-axis machines the 4th axis rotation is achieved by rotating the spindle motor and the cutter (swiveling head). DeskProto does no support such machines: only machines where the 4th axis rotation is achieved by rotating the material block.

This page is about continuous rotation, which is in fact 'real' **Rotation axis machining**. It can be switched on by checking the box "Use rotation axis" in the Part parameters. That option is of course available only when the machine that you selected for your project has a rotation axis configured in it's machine definition.

In the DeskProto calculations the rotation axis is always **parallel to the X-axis**. In case on your machine it is **parallel to Y** you can configure that in the machine definition: Options > Library of machines > OK on warning > select your machine and press Edit. One of the options in the Rotary Machine Settings is "Has 4th axis parallel to Y". This option will swap X and Y: more information on that help page.

Geometry data
In case only Geometry data is present DeskProto will set a Material block that exactly fits the bounding box of the geometry. This block may either be rectangular or cylinder shaped. The axis of this cylinder (so the actual axis of rotation) will be the X-axis **in the original 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.
Note the easy way that is available to create a rotary project for a geometry: using the Rotary machining wizard.

Vector data

For Vector data two different data types are possible: 2D Vector data (no Z-values, or all Z-values equal to 0.0) and 3D Vector data (Z-values are present that are not equal to 0.0). In the latter case DeskProto offers the option to ignore all Z-values, which converts the data to 2D.
In case only **2D Vector data** is present, rotation axis machining is possible only when a **cylinder** shape Material block has been set. DeskProto will create a cylinder 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 2D curve with Y-size 100 mm, the diameter of the generated cylinder block will be 31.83 mm: as this cylinder's circumference is Pi * 31.83 = 100 mm. You can use a larger diameter cylinder by setting the material block to custom. The machining depth is relative to the outside surface of the cylinder block.

In case **3D Vector data** is present DeskProto will offer two workflows:
1- **wrap** the curves round a cylinder (just as for 2D data), and use the Z-value as radius for each point on the curve. The machining depth then is relative to this Z-value.
2- **do not wrap**, so keep the 3D curves as they are, in 3D space. The toolpaths then will be calculated based on these curves, just as done in the calculation for rotary geometry toolpaths. This option will be useful for instance when you have cut a 3D shape and have a 3D curve over the surface (from the same CAD system) that indicates a groove in the surface: DeskProto then can use vector machining to cut the groove. The machining depth is relative to the radius value of each point of the curve.

When a vector file is loaded that contains 3D curves DeskProto will ask if you want to use the Z-values in the curves, and also if you want to to wrap or not to wrap in case rotary toolpaths are required.

Bitmap data
In case only Bitmap data is present DeskProto will also create a cylinder shape block and wrap the image round that cylinder. Exactly as described above for 2D vector data.

For **mixed projects** the Geometry size will overrule the wrapping calculation when calculating the block size.
For mixed projects you can select "Project on 3D Part geometry" in the Z-settings for the vector data and/or the bitmap data.machining. The machining depth / Relief depth then is taken relative to the geometry surface at that XY position.
In all cases you are of course free to change the block dimensions conform your needs.

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**

will move. 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, and as on some rotary machines Y-movements are not possible.

A wealth of information on rotation axis machining is available in the DeskProto Tutorial book (can 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 of Milo statue. This STL file is a free download on www.deskproto.com In addition a series of nice tutorial video's are available on the DeskProto website.

5.4.3 Indexed machining

A rotation axis (4th axis) on a CNC milling machine can be applied in two different ways:
- the part can be rotated during machining (continuous rotation)
- the part can be machined from one side (three axis machining), then rotated and machined from a second side, etc.

DeskProto supports both methods, obviously only in the Multi-Axis edition of DeskProto.

This page is about **indexed machining**, which is possible only for geometries. DeskProto offers indexed machining in wizard Advanced geometry, **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.

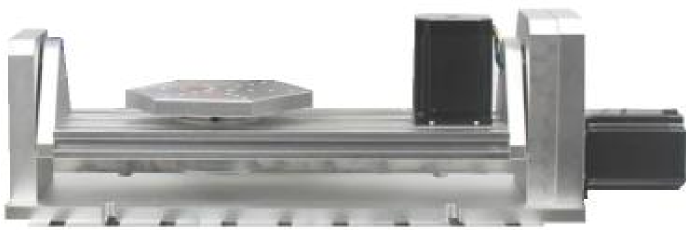
- The wizard will automatically generate N parts for you in the DeskProto project tree, and will apply the correct rotation for the geometry in each Part. The rotations are evenly spread (360 / N), if needed you can select a higher N and then later delete some of the parts.
- Support tabs (cylinder shaped blocks left and right), to keep the part connected to the rotary table, 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. The wizard will also use these commands to move the Z up to a safe height before rotating the block. This is needed because the diagonal of a rectangular block is larger than its width and height.

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 rotations for each part, and specify these same rotations 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. For five-axis machining these commands need to be added manually, as no 5-axis wirzard is present.

5.4.4 Five-axis machining

DeskProto offers a limited support for five-axis machines:
- only **indexed machining**, no continuous rotation or simultaneous 5-axis. This means no rotations while machining, only rotations in-between the 3-axis toolpaths for the various orientations of the part.
- only "**trunnion style**" 5-axis machines are supported: machines in which two rotary tables are combined that will rotate the block of material. See the illustration below, where a simple trunnion-style 5-axis unit is shown, positioned on a machining table with T-slots. The large rotation unit (the horizontal axis) is used as tilting table, the small rotation unit (in the image the vertical axis) can rotate the part. One more photo of such machine can be found on the rotary machine settings page.



DeskProto does **not** support machines where one or both rotations are achieved by rotating the spindle motor and the cutter (swiveling head machines). We do not plan to extend the 5-axis options in DeskProto, as the program needs to be easy to use: offering "*CNC machining for non-*

machinists". Five-axis machining is a complex process, which does not match with this primary aim.

The five-axis process is much more complex than for four axes, so DeskProto cannot offer a wizard to automatically create such project for you. It works the same as indexed machining with four axes: you create a series of parts, specify the correct rotations for each part, and specify these same rotations 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.

Very important is that the translation settings need to be None for all axes: **the zero point on the machine needs to be the same as in the CAD data**. On the machine this zero point needs to be set **exactly** on the point where both rotation axes intersect, as DeskProto will also perform all rotations round axes through this zero point.

These are the steps in the process:

- Define a part for each side that you want to machine, and apply the correct rotation for the geometry in each Part.
- Define the operations that you need for each part
- Set the rotation commands for A and B (to be done in-between the operations) by entering them in the operation's Start/End commands. The rotation values need to be the same as just set for each part. Also enter commands to move the Z up to a safe height before rotating the block.
- Link all operations into one large Chain in order to write one combined NC file for the complete indexed milling process.

It will be clear that this is only meant for experienced DeskProto users. 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.

5.4.5 Laser engraving

Many small CNC milling machines offer a laser unit as optional extra. This is a device that emits a downward laser beam on the part that will burn the material. It can be used to change the color and thus write/draw on the part (laser engraving), or to completely cut the part (laser cutting).

DeskProto supports using such laser for Vector machining (so not for creating 3D geometry parts or for bitmap images). This can be done in 2D on a flat piece of material, and also in 3D by projecting the vector curves on the 3D geometry of a part that has been machined earlier.

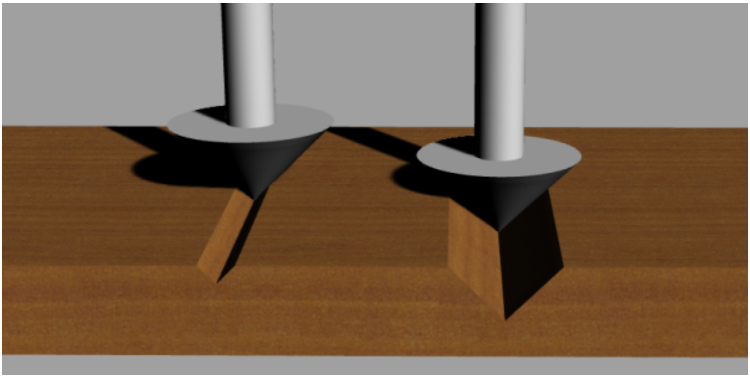
In order to use Laser engraving the following things need to be configured first:
1. In your postprocessor you need to enable the laser option (on tab page Laser)
2. In your machine definition you need to enable the laser option (in the machine's Laser settings).
3. You need to define one of more laser cutters.

After that you will be able to select one of these laser cutters in the Vector operation parameters.
This will change the following options in the settings that this dialog offers:
- the word "Spindlespeed" is replaced by "S-value", with the machine's laser power values as min and max. That is because many machines use the S command (in G-code) to set the laser power.
- an option "Pass count" is added, to be used when your laser is too weak to get the desired result in one pass.
- the Machining depth is disabled, at 0 mm: this is about engraving on the surface of the material.
- Peck drilling is disabled
- all Roughing options are disabled
- the Dynamic feedrate is disabled.

The toolpaths will be different as well:
- During rapid positioning movements the laser will be switched off
- The toolpath needs to go up and make a rapid movement for every positioning movement (for very small movements a milling cutter may remain at machining level) We call this behavior "Never stay low".

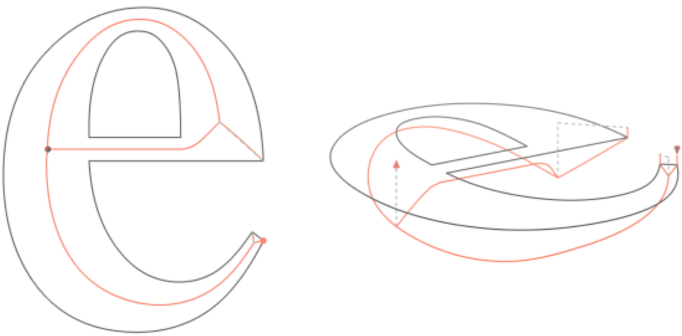
And the Z=0 position needs to be set at the level where the laser's focus hits the material.

5.4.6 V-Carving



V-Carving is a special type of Vector machining. You can find the settings on the V-Carving tab of the Vector operation parameters. For this toolpath type a conical cutter needs to be used, also called "V-Cutter" or " V-bit". The trick with such cutter is that the width of the groove that is machined depends on the depth of the cut.

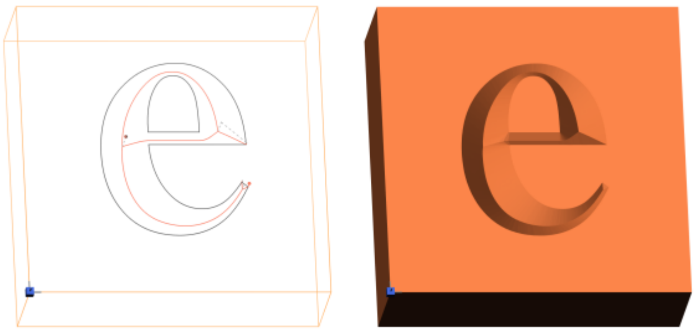
Assume that you have a 45 degree conical cutter (included angle 90 degrees) and that the Z=0 is set with the tip of this cutter (a sharp tip) at the top surface of the material. The image above shows how a toolpath at Z=-1 mm results in a groove that is 2 mm wide and a toolpath at Z=-3 mm will make a 6 mm groove.



You can imagine that a 3D toolpath with a varying depth will create a groove with a varying width. **V-Carving** needs closed vector curves: it calculates toolpaths with a depth that matches the width of the pocket at each position.

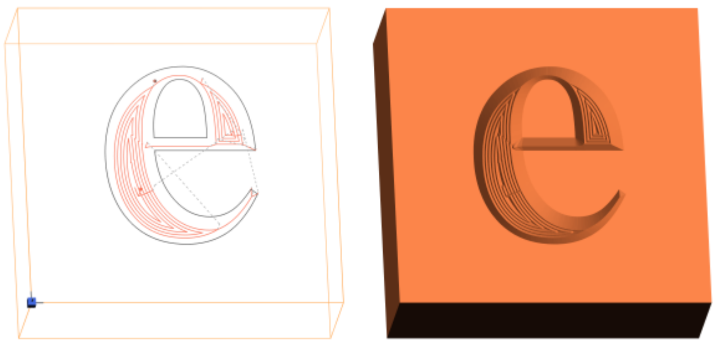
It is also possible to create sharp corners in the resulting groove: using a toolpath that rises to the surface (Z=0) in a movement at a certain angle.

V-Carving works very well to machine a text in a detailed font: the screenshots above show the V-Carving toolpath for an "e" character, in font Times New Roman (with a 45 degree conical cutter). The varying depth makes sure that the width of the groove will exactly match the outline of the drawing. Note the short toolpaths on the right side towards the three sharp inner corners.



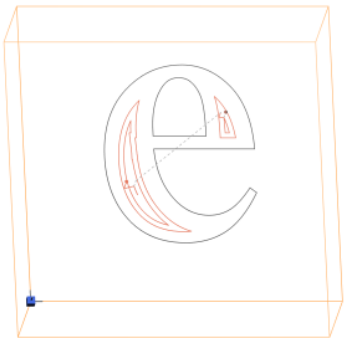
The two images above show the toolpath and, from the same viewpoint, a simulation of the resulting part. As you can see: the outside shape of the groove indeed exactly matches the vector curves of this font.

It will be clear that a V-Carving toolpath will go deep when when the groove is wide and/or when the V-cutter has a sharp point (small angle). When needed the toolpath will go down in steps (roughing layers). DeskProto will show you how deep the unrestricted toolpath will dive (after calculating the toolpaths, on tab page V-Carving).



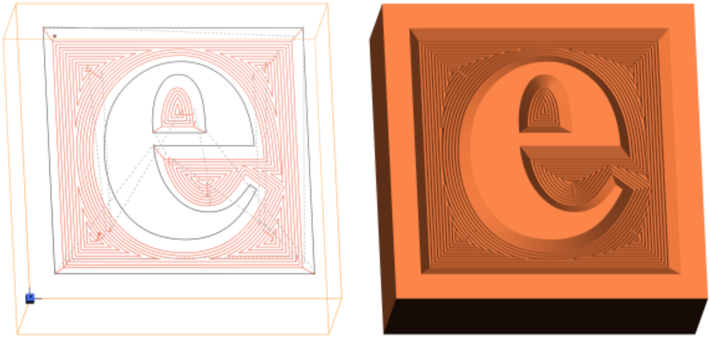
In case you do not want to cut to this unrestricted depth DeskProto offers you the option to limit that depth to the Machining depth that you have set (Vector operation parameters, tab Z Settings). All cutter movements below this machining depth will be skipped, and DeskProto will fill the resulting horizontal surface with pocketing toolpaths.
When you do not limit the depth it will still not be completely unrestricted: the V-Caving toolpaths will not go below the bottom of the block.

In the image above you can see both the toolpaths and the result. The V-cutter that is used has a sharp tip, so it is clear that this bottom surface will not be flat, as you can clearly see in the simulation on the right..



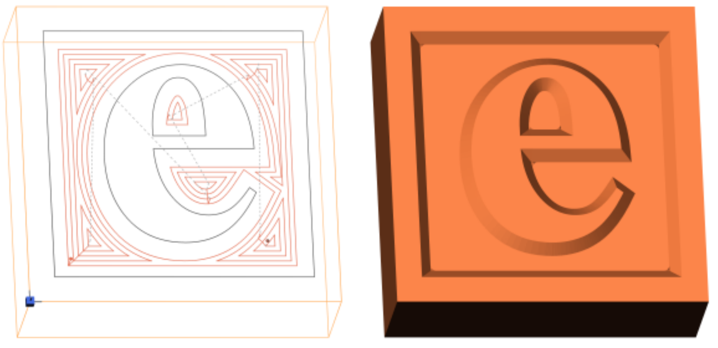

It is possible to make this bottom flat by adding an extra operation: a **Cleanup operation**. You can easily add such operation using button "Create cleanup operation..." that you can find on tab V-carving of the Operation parameters. That extra operation uses the same curves and the same machining depth as the V-Carving operation, however it uses a flat cutter (instead of a V-cutter) and Pocketing toolpaths. It also contains one special setting: the XY allowance, set on the Roughing tab, needed as the flat cutter needs to keep a certain distance away from the curves. This distance depends on the depth of cut and the angle of the cutter, as explained on the Cleanup operation help page just mentioned. For a 45 degree conical cutter this is easy: the allowance must be exactly as large as the machining depth.

As you can see in the image above: not the complete bottom will be made flat. The end mill (flat cutter) cannot reach into sharp inner corners any further than its radius allows.



The result of a V-Carving operation can be completely changed by adding one closed vector curve around the complete vector drawing. Like in the image above the square around the "e". When you now (again) select All curves the background of this "e" will be V-Carved, as shown above. Which again is a great result !

Obviously this will take longer as the toolpath now is much longer.



Here as well you can add a Cleanup operation to make the bottom surface flat.

As said this toolpath type is a great tool for creating signs: you will be amazed how quickly you can machine an absolutely good-looking sign using V-Carving. It can of course also be used for graphic designs other than just text. It can also very well be used for Inlays.

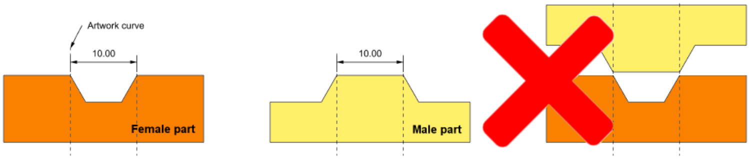
5.4.7 Inlays (wood-in-wood)

A wood-in-wood inlay uses two kinds of wood, in different colors, to achieve a part with a graphic image on its top surface. The photo above shows a nice inlay of a rose icon. The image can of course also be a text.

Creating a wood inlay involves cutting a pocket in the flat top surface of the **base** (the female part), and cutting a part that will exactly fit in that pocket (the male part, also called the **plug**). The plug then is glued on its position inside the pocket, and finally the top surface is finished to make it smooth and even.

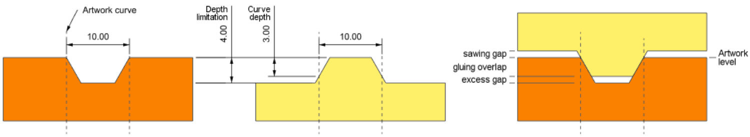
We have found that V-Carving is a great way to create perfect inlays. When using a V-bit, both the base and the plug will have side walls at a certain angle, which makes the gluing process very easy. All male pieces to be inserted are present on one large part. When gluing, the two parts simply are guided to their correct position, and the process is forgiving when they are not perfectly aligned. Due to these angled walls this type of inlay is also called a **prismatic inlay**.
The other important advantage is that due to the V-bit's sharp point it is possible to create inlays with sharp corners.

How it is done



For the **male part** two extra settings are needed:
1. the artwork needs to be **mirrored**, as the plug will be used upside-down.
2. an **extra curve** needs to be selected: a rectangle around the complete design. As a result all material between the artwork curves and that rectangle will be removed.

The schematic drawing above shows what happens when you simply machine both the male part and the female part with the same V-Carving settings. The design is a groove of 10 mm wide, and the cutting depth is limited to 4 mm. In both case the artwork is positioned on the top of the material block.It will be immediately clear that the plug (the male part) will not fit in the base (the female part). So an extra setting is needed: the curve depth.



The **female part** was already correct: the artwork curves are present at the top surface, which will be the surface that is visible in the result. So the result will show the original artwork.

For the **male part** we used this extra setting, and entered a **Curve depth** of 3 mm. The cutting depth limitation still is 4 mm, just as for the female part. In the drawing you can see what happens when the artwork curves are located 3 mm below the top of the block: the plug now nicely fits in the base, with 3 mm overlap.

On both sides a gap of 1 mm remains:
- the **sawing gap** is used to easily saw off the remainder of the plug
- the **excess gap** provides room for an excess of glue
With in-between the **gluing overlap**, where both parts will be touch (under pressure while gluing) to make a tight connection.
Both gaps also provide room for the remaining cusps on the horizontal surfaces: the V-bit cutter with a sharp tip cannot make these surfaces flat, so in-between the toolpaths cusps will remain (unless you also use a cleanup operation).

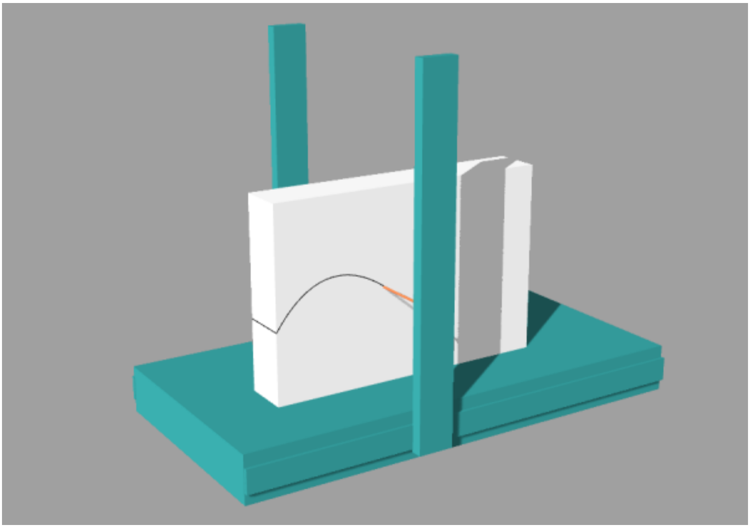
An excess gap is not absolutely needed: you can also machine the female part without depth limitation (the block needs to be thick enough), and for the male part use a curve depth that is equal to the unrestricted depth of the female part. As depth limitation for the male part use a Z-depth that is a bit lower: to create the sawing gap.

The image above shows the two parts for the rose inlay, just before gluing. As both parts have been machined using the same cutter the angle of the side walls is identical. The shapes on the male part (the part on the left) may seem a bit small: that is because the artwork is positioned 3 mm below the top of the block. So the shapes that you see on the top of this block are indeed smaller than in the artwork.

The exact depth values to be used for creating an inlay will depend on the size of your inlay and the type of wood. The values that are shown on this page anyway provide a good start.

The male part (the plug) typically has a large area where all material needs to be removed (up to the depth limitation). The V-bit cutter is not an efficient tool to machine such large horizontal area. You can add a **Pocketing operation** with a flat endmill to flatten that area, as explained on page V-Carving. For that pocketing operation you then need to set an **Allowance** (Roughing tab), as the flat cutter needs to stay away from the artwork curve in order not to remove the lowest part of the angled side walls that the V-bit machined.

5.4.8 Hot Wire cutting



A hot wire cutter is a machine for cutting PS foam (PolyStyrene) It uses a thin metal wire that is heated and can be used to create a thin cut through a block of EPS by melting the material. On CNC machines the path of the hot wire is CNC controlled. The wire on such machine always is a straight line, in most cases horizontal.

The image above shows a very basic image of such machine: the orange line is the hot wire. This machine can control the Z-position of the wire as that can move up and down along the two columns, The columns can move to the left and to the right to control the X position: so each end of the wire can make a 2D movement (XZ). DeskProto creates hot wire toolpaths where both ends of the wire (both columns) make the same 2D movement.

Clients who CNC machine large parts in EPS foam can make their production process much more efficient by combining CNC machining with hot wire cutting, using either two separate machines or one machine that combine both technologies. They remove as much PS foam as possible using a hot wire, which is faster than by machining, and greatly reduces the amount of chips and dust that is created. This works best when combined with a rotation axis, as then the hot wire can make a number of cuts, at different angles. The finishing then can be done using a milling cutter, with standard DeskProto toolpaths.

Configuration

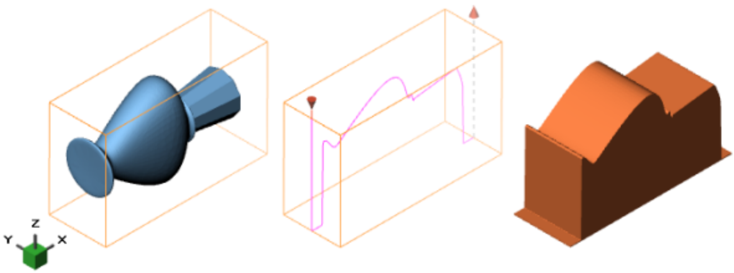
DeskProto supports using such hot wire for Geometry machining and for Bitmap machining (so not for vector data). Though it will be applied primarily for rotary machining of geometry data.

In order to use hot wire cutting the following things need to be configured first:
1. In your postprocessor you need to enable and configure the hot wire option, see below (on tab page Hot wire).
2. In your machine definition you need to enable the hot wire option (in the machine's Hot wire settings).
3. You need to define one or more hot wire cutters.

Project settings

After that you will be able to select one of these hot wire cutters in the Operation parameters.
This will change the following options in the settings that this dialog offers:
- for 3-axis machining the Distance between toolpaths is disabled, as then only one toolpath is generated
- for 4-axis machining the pre-set values for Distance between toolpaths are different, see below
- the Spindlespeed setting is disabled
- most Strategies are disabled: only parallel toolpaths along X are possible
- most Roughing options are disabled
- the Dynamic feedrate is disabled.

The toolpaths will be different as well:



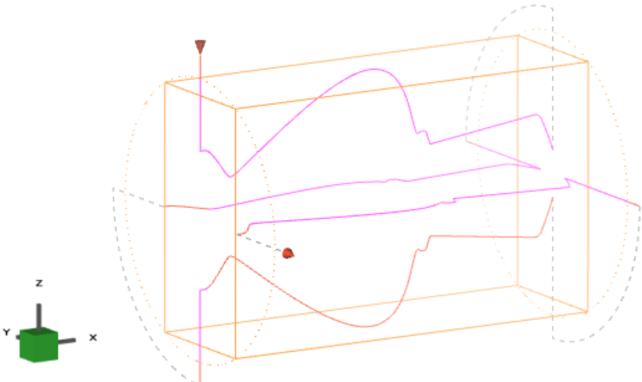
Here is an example DeskProto project, featuring sample geometry Bottle.stl and a hot wire cutter with a 1.5 mm diameter (for a hot wire cutter this is not the diameter of the wire but the width of that path that is melted). From left to right you see the geometry, the toolpath (strategy Parallel along X, only X and Z are used) and the simulation. Only one toolpath will be generated, as this cutter is assumed to be long enough to cut the complete block in one path (in the cutter definition this is called "Auto length"). The toolpath is **two-dimensional**, and when writing the NC file DeskProto will suppress the third coordinate (in this case the Y). That is, unless in the postprocessor you have defined that it is a required coordinate.

When you select a hot wire cutter you will see that only **Parallel along X** is available as Strategy, which makes sense for this one toolpath. In addition the **Roughing layers will be disabled**: after cuttings this toolpath the excess material needs to be removed by hand. When using a milling cutter all that material would need to be converted to chips, using roughing layers. Ramping is disabled as well, as that would create unwanted extra movements. Of the two precision settings only the **Stepsize** can be set, as a Toolpath distance is not present. The Stepsize needs to have a low value, as the diameter of a hot wire cutter (the widths of the groove) is small. That is not a problem as DeskProto will filter unneeded points from the toolpath.

The simulation in the above image shows a limitation of using a hot wire for roughing: at the curved surface to the front side and to the back side of the bottle no material has been removed. So it may well be that in the first operation with a milling cutter it is needed to use roughing layers. DeskProto will not do so automatically (it will warn you though).

Using a rotation axis

The 2D toolpaths as described above are pretty standard for hot wire cutting: any user of such machine will already have software that can create this type of toolpaths. The situation is different when the machine also has a CNC controlled rotation axis: this is where DeskProto stands out, and in practice rotary cutting is the main use of DeskProto's hot wire option.



In order to use this you simply check the option "Use rotation axis" in the Part parameters. In the Geometry Operation parameters you select a hot wire cutter. You will see that now you can again set the Distance between toolpaths (which for the three-axis toolpaths as described above was not possible). The values that you can choose from now are not based on the cutter diameter (as they normally are), but on the circumference of a cylinder around the part. Simply ignore the actual value and look at the **denominator of the fraction** that is shown. For instance option "c/4" will make DeskProto calculate 4 toolpaths, evenly spaced around the rotation axis, so at 90 degree angles of the A-axis. The image above shows four rotary hot wire toolpaths for the bottle geometry.

Postprocessor

The NC file that DeskProto will create for hot wire toolpaths (using a G-code based postprocessor) will include coordinate values for X, for Z and (for rotary toolpaths) for A. However, we have seen that some hot wire cutting machines use different coordinates. For instance: for a machine as shown on the top of this page some controllers use X and Y (Y for the height of the wire), instead of the X and Z that DeskProto uses. For such machines Z is the length of the wire. Other machines use A, B or C to control the hot wire. To accommodate such machines you can configure this in the postprocessor.

As the hot wire toolpath is a 2D toolpath DeskProto will automatically suppress the third coordinate (column Y). Note that you should not make this coordinate required (by marking it with a + in field Order/Required). A and B coordinates are written only when needed.

5.5 Various

5.5.1 Sample files

Every DeskProto installation will put a number of sample files on your computer, both CAD files and DeskProto projects. These allow you to immediately start working with DeskProto, even if you do not yet have CAD data to load. They are also used in the DeskProto tutorials (PDF book and video tutorials). In addition a number of larger CAD data files can be found on the DeskProto website, as free downloads.

The sample project files (DPJ files) have been prepared to be used as sample by removing the following settings:
- the path to the CAD data file(s)
- the machine to be used
- the feedrate and the spindlespeed

When DeskProto opens such prepared sample project it will load the CAD data from the current folder (which is the Samples folder), it will use the machine that you have set as your default machine, and it will use the default speeds for that machine.

The sample files are copied to the location that your operation system prescribes for such files, making it difficult to find them (sorry). Quickest way to find the sample files is via the DeskProto Start Screen: simply check "Use samples folder" on this screen. The Sample file location is:
Windows: *C:\ProgramData\DeskProto 8.0\Samples*
MacOS: *~/Library/Application Support/Delft Spline Systems/DeskProto/8.0/Samples/*
Linux: *~/.local/share/Delft Spline Systems/DeskProto/8.0/Samples/*

The following sample files have been installed with DeskProto (for most samples a metric version and an inch version):

A nice **Beer tray**, files:
2D_DpBeerTray.dxf DpBeerTray.dpj DpBeerTray_FreeEdition.dpj
2D_DpBeerTray_inch.dxf DpBeerTray_inch.dpj DpBeerTray_FreeEdition-inch.dpj

A small perfume **Bottle**, files:
Bottle.stl Bottle.dpj Bottle_FreeEdition.dpj Bottle-RotAxis.dpj Bottle-withShellRelief.dpj
Bottle_inch.stl Bottle_inch.dpj Bottle_FreeEdition_inch.dpj Bottle-RotAxis_inch.dpj Bottle-withShellRelief_inch.dpj

A very simple metal **Casting**, files:
Casting.stl Casting.dpj

Casting_inch.stl Casting_inch.dpj

The **DeskProto logo** as vector data, files:
2D_DeskProtoLogo.dxf DeskProtoLogo.dpj DeskProtoLogo_inch.dpj

A **Die**, as sample project for **5-axis** machining, files:
Die-5axisVideo.stl Die-5axisVideo_inch.stl Die-5axis.dpj Die-5axis_inch.dpj
Take care: these settings are for a specific machine, for your machine you will need to change them.

A very simple **Flat test** surface (can be used for toolpaths to flatten the top of your block), files:
Flattest.stl Flattest.dpj Flattest_inch.dpj

Set of custom **Insoles** (only in mm), files:
Insoles.stl Insoles.dpj

A nice **Picture frame**, files:
DpPictureFrame.stl DpPictureFrame.dpj
DpPictureFrame_inch.stl DpPictureFrame_inch.dpj

A **Ring**, as sample jewelry wax model, files:
SampleRing.stl SampleRing.dpj

A simple **Sphere**, files:
Sphere.stl Sphere.dpj
Sphere_inch.stl Sphere_inch.dpj

A human **Torso** (work of art), files:
Torso.stl Torso.dpj Torso-FourSides.dpj
Torso_inch.stl Torso_inch.dpj Torso-FourSides_inch.dpj

The **Utah teapot**, famous in computer graphics history, files:
Teapot.stl Teapot.dpj
Teapot_inch.stl Teapot_inch.dpj

A **V-Carving** project, with parts for for engraved text and for elevated text.
2D_TimesNewRoman_200x30mm.dxf V-Carving_TimesNewRoman-mm.dpj V-Carving_TimesNewRoman-inch.dpj

As simple bitmap project the **XYZ logo**, files
XYZlogo.png XYZlogo.dpj XYZlogo_inch.dpj

Various **Vector files**:
2D_Square.dxf 2D_Square_inch.dxf 2D-CircleD5.ai 2D-CircleD5.eps 3D_Polyline_Helix.dxf

Various **Geometry files**:
Cube10x10x10.stl

Various **Bitmap files**:
Radetzky.jpg Shell1.jpg Shell2.jpg
Radetzky.dpj

5.5.2 Editions / Trial mode

DeskProto is offered to you as '**Freemium**' software:
you are welcome to use the *basic functionality* DeskProto **free** of charge in the Free edition, the *advanced features* are available as **premium** extras in the three paid editions.

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 can be used as free CAM software, as long as you like. As said: it offers only basic CAM functionality. It also allows you to **Trial** (evaluate) the higher editions; when running in **trial mode** the resulting toolpath will add a **Trial cross** (watermark), visible 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.
Compared to the Free edition a few important extra options are present: Pocketing, Drilling and V-Carving for Vector operations, and Roughing for all operation types. Projects also may contain any number of parts and operations.

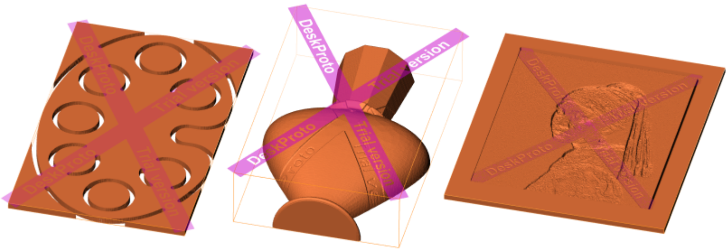
The **Expert Edition** includes all parameters except for the rotation axis options. So the fourth axis and the fifth axis are not available in this edition.

The **Multi-Axis Edition** is the most complete version: all parameters are present, including support for 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 paid edition can be activated in the Activate dialog.
After activating your license this Edition select dialog will no longer be displayed.

5.5.3 Trial Cross



The Free edition allows you to **Trial** (evaluate) the higher Editions: when running in trial mode the resulting toolpaths 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 images:
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 made a bit lower, 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 parts will suit your needs, however the trial cross will make really using the parts 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.

5.5.4 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. Reason is that DeskProto uses absolute rotations: for each new orientation DeskProto starts with the original CAD orientation, first applies the X-rotation, next the Y-rotation and finally the Z-rotation.

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) as 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 to find the orientation that you need is the option called Show downward faces in the Items visible dialog.

5.5.5 Postprocessor

The postprocessor is the part of the software that 'formats' the NC file. So it translates the toolpath information into the format that your machine can understand. Using the word 'postprocessor' is common in the world of CAM software and CNC. In the world of computers and printers you might call this a 'device driver'.

```
%
G00 X0.000 Y0.000 Z5.000 S2000
G01 Z-1.000 F1000
G01 X100.000
G01 Y100.000
G01 X0.000
G01 Y0.000
G00 Z5.000
M30
```

```
N1IMF_PBL
N5 FASTVEL 50000
N6 SPINDLE ON
N7 FASTABS X0 Y0 Z5000
N8 VEL 25000
N9 MOVEABS X0 Y0 Z-1000
N10 MOVEABS X100000 Y0 Z-1000
N11 MOVEABS X100000 Y100000 Z-1000
N12 MOVEABS X0 Y100000 Z-1000
N13 MOVEABS X0 Y0 Z-1000
N14 FASTABS X0 Y0 Z5000
N15 SPINDLE OFF
N16 PROGEND
```

```
^DF
V22
Z 0,0,500
V8
Z 0,0,-100
Z 10000,0,-100
Z 10000,10000,-100
Z 0,10000,-100
Z 0,0,-100
V22
Z 0,0,500
H
```

The image above shows exactly the same toolpath in three different NC formats. It is a simple square of 100 x 100 mm, 1 mm below zero. Left you see standard G-code, in the middle a proprietary format of a large machine manufacturer, right a type of plotter control language. The postprocessor determines the format that will be used when writing an NC file.

Important to know is that DeskProto does not use one unique postprocessor for every machine (like some other CAM programs do). In practice you will see that many different machines can work using the same NC file format. For instance a series of machines by one manufacturer that only change in size. Also: machines from various manufacturers can use the same controller (and/or the same control software, like for instance Mach3). So it makes sense to make different definitions for the machine and for the postprocessor, which is what DeskProto does. **Various machines then can share the same postprocessor**: for instance more than 20 machines (by various manufacturers) are installed that use the Mach3 postprocessor.

In DeskProto the machine definitions can be configred in the Library of Machines.
and the postprocessors can be configured in the Library of Postprocessors.
One of the settings in the machine definition is which postprocessor needs to be used.

5.5.6 Postprocessor placeholders

The option to use Placeholders is a very powerful feature in DeskProto. A placeholder is a bit of text that you enter, which when the NC file is written will be replaced by the current value of some parameter.
An example: you can enter {DATE} , which in the NC file will be replaced by the current date.

In DeskProto placeholders are enclosed in curly braces (curly brackets). They can be used in the Postprocessor definition and in the operation Start/End commands

Some of the placeholders can be used in code that is processed by the machine, like Feedrate and Spindlespeed. Other placeholders can only be used in comment lines: these can of course be used only if your machine or your control software supports comment lines (for instance lines that start with a semicolon, or all code between parentheses). Consult your machine's manual to find out.

This is the list of placeholders that can be used in DeskProto V8.0:

Project information:

`{PROJPATH}`	The complete file specification of the project file (.dpj), including disk and path
`{PROJNAME}`	Name of the project
`{MACHINENAME}`	Name of the machine that has been selected for this project
`{POSTPROCNAME}`	Name of the postprocessor that is used (so that is configured for this machine)

Part information:

`{PARTNAME}`	Name of the current part
`{BLOCKXSIZE}`	Size of the material block along X. Sizes are given in the units as defined in the Preferences, used in DeskProto's user interface.
`{BLOCKYSIZE}`	Size of the material block along Y
`{BLOCKZSIZE}`	Size of the material block along Z
`{BLOCKDSIZE}`	Diameter of a cylinder shaped block (only for parts with rotation axis toolpaths: no output for 3-axis parts).
`{BLOCKRSIZE}`	Rmax minus Rmin, for a cylinder shaped block. For a tube this is the material thickness., for a solid cylinder the Radius (only for parts with rotation axis toolpaths: no output for 3-axis parts).
`{BLOCKXSIZE_IN_NCFILE_UNIT}`	Size of the material block along X. Sizes are given in the units as defined in the Postprocessor to be used in the NC file.
`{BLOCKYSIZE_IN_NCFILE_UNIT}`	Size of the material block along Y
`{BLOCKZSIZE_IN_NCFILE_UNIT}`	Size of the material block along Z
`{BLOCKDSIZE_IN_NCFILE_UNIT}`	Diameter of a cylinder shaped block (see above).
`{BLOCKRSIZE_IN_NCFILE_UNIT}`	Rmax minus Rmin, for a cylinder shaped block (see above).
`{PARTMACHTIME}`	Estimated machining time for the part

Operation information:

{OPERNAME}	Name of the operation to which the current toolpath belongs
{TOOLNUMBER}	Number of the current cutter (as set in the cutter definition)
{TOOLNAME}	Name of the current cutter
{FEEDRATE}	Feedrate, in the units as defined in the Preferences, used in DeskProto's user interface. Only the number if written, without the Feedrate command (like "F")
{FEEDRATE_IN_NCFILE_UNIT}	Feedrate, in the units as defined in the Postprocessor to be used in the NC file.
{F}	Same as line above
{SPINDLESPEED}	Spindlespeed - only the number if written, without the Spindle command (like "S")
{S}	Same as line above
{OPERMACHTIME)	Estimated machining time for the operation.

NC file relevant information:

{PROGNUMBER}	Number of the NC program: some controllers require a unique program number as identification.
{TOOLPATHXMIN}	Minimum X-value in the toolpath, in user-interface coordinates, as set in the Preferences.
{TOOLPATHXMAX}	Maximum X-value in the toolpath.
{TOOLPATHYMIN}	Minimum Y-value in the toolpath,
{TOOLPATHYMAX}	Maximum Y-value in the toolpath.
{TOOLPATHZMIN}	Minimum Z-value in the toolpath,
{TOOLPATHZMAX}	Maximum Z-value in the toolpath.
{TOOLPATHXMIN_IN_NCFILE_UNIT}	Minimum X-value in the toolpath, in NC-file coordinates, as set in the Postprocessor.
{TOOLPATHXMAX_IN_NCFILE_UNIT}	Maximum X-value in the toolpath.

`{TOOLPATHYMIN_IN_NCFILE_UNIT}`	Minimum Y-value in the toolpath,
`{TOOLPATHYMAX_IN_NCFILE_UNIT}`	Maximum Y-value in the toolpath.
`{TOOLPATHZMIN_IN_NCFILE_UNIT}`	Minimum Z-value in the toolpath,
`{TOOLPATHZMAX}`	Maximum Z-value in the toolpath.

Other relevant information:

`{N}`	Newline: forces DeskProto to continue on a new line
`{DATETIME}`	Date and time, for instance "2021-02-12 14:07"
`{DATE}`	Date, for instance "2021-02-12"
`{TIME}`	Time, for instance "14:07"
`{{}` or `{}}`	This is the trick if you need a curly bracket to appear in your NC file.

The application of the above placeholders will be clear, except perhaps for the Program number: DeskProto can't know which program number you want to use so it cannot replace this placeholder on the fly.
So when you have used the placeholder `{PROGNUMBER}` in your project and then write an NC file, the NC-Program File Number dialog will pop up, asking you to enter the program number that you want to use for this NC program. For many machines this number need to be preceded by a capital "O": do not forget to write that just before the placeholder.

The PROGNUMBER placeholder needs to be placed in the Start commands of the postprocessor, as this program number is needed at the start of the program. When you use this placeholder in the Operation Start/End commands the NC Program File Number dialog will not pop up.

Example:
So the Start commands of your G-code postprocessor can for instance be:

```
%
O{PROGNUMBER}
; NC program written by DeskProto on {DATE},
; for machine "{MACHINENAME}".
```

```
; Start with cutter "{TOOLNAME}".
```

The resulting G-code then would start with something like

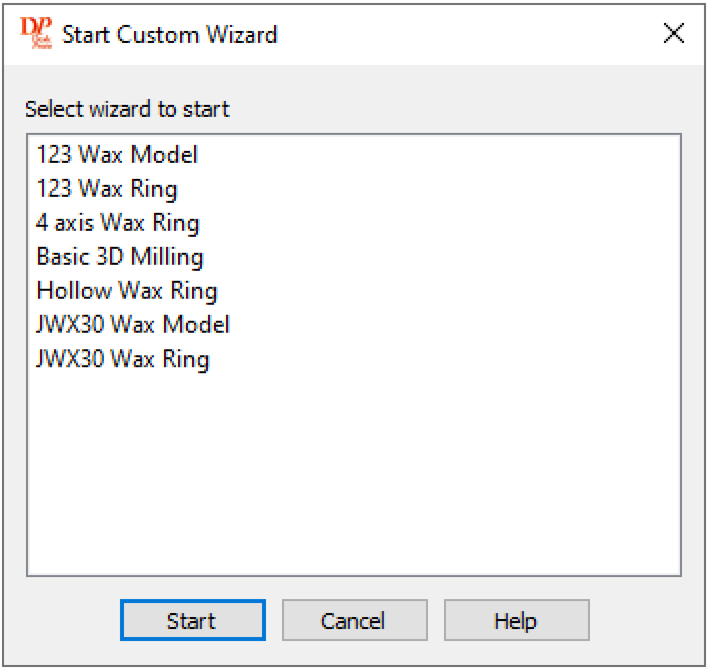
```
%
O1234
; NC program written by DeskProto on 2021-02-12,
; for machine "ISO plain G-codes - mm".
; Start with cutter "Ball nose, radius 3 = diameter 6
mm".
```

Upwards compatibility.
The use placeholders has been added in DeskProto V7.1
Previous DeskProto versions offered only two optional placeholders in the postprocessor, using a different format:
- the string "`^\N`" started a new line,
- the string "`^\V`" would again output the parameters value of that tab page (for instance the Spindlespeed).

These old placeholders are no longer supported in DeskProto V8 When opening an old postprocessor file DeskProto V8 will automatically convert these to the new placeholders.
For instance a Start command "` S ^\V^\N R`" on the Spindle tab page will be converted to "` S {S}{N} R`". The postprocessor file will be saved in the V8 format only when other changes have been entered.
A postprocessor file in the new V8 format can no longer be used in older DeskProto versions.

The conversion and compatibility just described also applies to DPJ files (DeskProto project files) that use "`^\N`" in the Operation Start/End commands.

5.5.7 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 have been installed, these can be started via File menu command Start Custom wizard.
You can also start a Custom Wizard by calling it's DPW-file as a command line parameter.

Custom wizards are not available in MacOS and in Linux: only in Windows, sorry about that.

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 four wax ring wizards that come with DeskProto

123 Ring	Wax	for the 123WaxRing fixture
4 axis Ring	Wax	for any machine with a 4th axis
Hollow Ring	Wax	for the 5th axis of the Roland JWX-10 machine
JWX30 Ring	Wax	for the Roland JWX-30 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. The custom wizards shown in the image above are already present, and can be found in folder
C:\ProgramData\DeskProto 8.0\CustomWizards
For instance the file Basic3D.dpw and the subfolder Basic3D

The **DPW file** specifies the name of the Wizard, how many pages are present, the page definition file for each page, version information. The page definition files are QML files, present in the folder just mentioned. For instance the file that shows the Load geometry page in the Basic3D custom wizard is
C:\ProgramData\DeskProto 8.0\CustomWizards\Basic3D\WizPageLoadGeometry.qml

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. 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. About QML and about JavaScript plenty of information is available via the Internet.

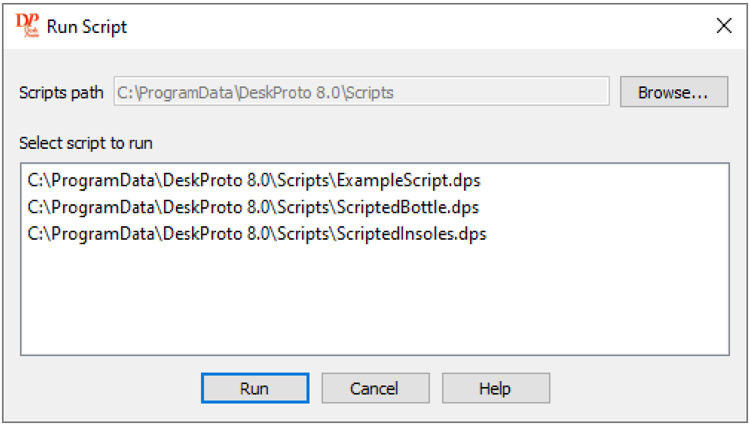
So basically all you need to add a Custom wizard to DeskProto is a plain text editor. Copy the resulting files to the locations specified above, and DeskProto will show your new wizard.

For more information on scripting see the Scripts page and the **DeskProto Script Documentation:** email us to receive a copy. Remember that an 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.

For software specialists:
you can set the location of this custom wizard directory as follows: in the Registry editor change registry key
HKCU\Software\Delft Spline Systems\DeskProto\8.0\Preferences\File Locations\WizardsLocation
Only edit the registry when you are qualified and know exactly what you are doing !

5.5.8 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), with 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, which will show this dialog.
You can also start a Script by calling it as Command line parameter.

Default folder for script files is the Scripts subdirectory of DeskProto:
Windows: *C:\ProgramData\DeskProto 8.0\Scripts*
MacOS: *~/Library/Application Support/Delft Spline Systems/DeskProto/8.0/Scripts/*
Linux : *~/.local/share/Delft Spline Systems/DeskProto/8.0/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 its 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):

```
MinDeskProtoVersion = 7.1;
MinDeskProtoRevision = 7828;
var strSamplesLocation = DPPreferences.getSamplesLocation();

DPProject.loadGeometryFile(strSamplesLocation + "Bottle.stl");
DPActivePart.setGeometryRotation(-90.0, 0.0, 0.0);
DPActivePart.materialBlockMethod = 2;
DPProject.calculateToolpaths();
DPProject.writeNCProgram( "ScriptOutputNCfile.ext" );
DPProgram.exit();
```

This script will load the Bottle geometry, orientate it correctly for machining, set the Material block 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 parameter of function writeNCProgram does not include a path: the file now will be save in the folder that is specified in the Preferences (tab NC Output). The file extension defined in this parameter will overrule that extension that is set in the postprocessor.

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.

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 as follows:
For Windows change registry key HKCU\Software\Delft Spline Systems\DeskProto\8.0\Preferences\File Locations\ScriptsLocation

For MacOS change line "8·0.Preferences.File Locations.ScriptsLocation" in file DeskProto.plist
For Linux change line "Preferences\File%20Locations\ScriptsLocation" in section "[8.0]" of file DeskProto.conf.
Only edit the registry or these files when you are qualified and know exactly what you are doing !

5.5.9 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 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.
In addition two extra parameters are possible, see below

1. First parameter:

Supported file-types for the first parameter are:

DPJ
This is a **DeskProto project file** - the project will be automatically loaded. For instance:
"C:\Program Files\DeskProto 8.0\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 8.0\deskproto.exe" "C:\Program Files\DeskProto 8.0\Wizards\123waxRing.dpw"

DPS
This is a **DeskProto Script file** - the script will be automatically started. For instance:
"C:\Program Files\DeskProto 8.0\deskproto.exe" "C:\Program Files\DeskProto 8.0\Scripts\ScriptedBottle.dps"

DXF, AI, EPS, SVG
These are CAD-files containing **Vector data** - the vector file will be automatically loaded. For instance
"C:\Program Files\DeskProto 8.0\deskproto.exe" "C:\ProgramData\DeskProto 8.0\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, 3MF, DXF
These are CAD-files containing **Geometry data** - the geometry file will be automatically loaded. For instance
"C:\Program Files\DeskProto 8.0\deskproto.exe" "C:\User\STLdata\medaillon.stl"
The default project needs to be a Geometry project.
Note that DXF can contain Vector data and/or Geometry data. The default project type decides how the file is loaded.

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 8.0\deskproto.exe" "C:\ProgramData\DeskProto 8.0\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 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 8.0\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.

This means that DeskProto can be included in an application without bothering the end-user about any DeskProto setting. When the end-user presses the "NC button" the application just needs to start DeskProto with the correct command line parameters. The result will be an NC file that the application then can send to the machine.

Extra parameters:

Two extra parameters are supported that are no file specifications. These flags may be added on any position on the command line, and do not count as first or second parameter.

"**/nocalculationwarnings**" will make DeskProto suppress calculation warnings.
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.*"

"**/userinteractionsuppressed**" will make DeskProto suppress all user interaction (except for error situations). This option includes the /nocalculationwarnings setting. Now also suppressed are the wizard interface, the chain warning, the dialogs for Load CAD data and Transform CAD data.

As said above, the first command line parameter may 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, but now for more than one set of default settings.
For instance:
"C:\Program Files\DeskProto 8.0\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-made 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 project causes 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.

5.5.10 Template projects

Some applications need 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 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: Yes. 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 may want to define template projects that you can use for various materials. 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 may want to define a template project for molds, using inverse milling.
- You may want to define a template project with two parts, for two halves of one model: one part for the left halve, the other part for the right halve (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.

Index

C

D

M

N

O

P

Q

R

S

W

Z