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 5.0

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

Table of Contents

I Introducing DeskProto

1.1 Disclaimer

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

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

1.2 Essentials

What does it offer
DeskProto is meant for any company that needs in-house facilities for Rapid Prototyping. Stereolithography machines and other Layer oriented RP systems are far too expensive, however a small CNC milling machine combined with DeskProto is affordable: offering **Desk**top **Proto**typing.
The advantages are clear: you no longer have to wait a number of days, your prototype is ready *within a few hours* ! The design process will be clearly accelerated.

How does it work
Starting point for DeskProto is a file with geometry from any 3D CAD-system. DeskProto reads the file and displays its contents. At this point it is possible to set parameters that describe the model (like the scale, rotation and the translation on the machine, etc). After entering parameters for the milling process (type of cutter, required accuracy, etc) DeskProto will fully automatically calculate the milling paths (called toolpaths). No danger for damaging the model as these paths are gouge-free ! After viewing the milling paths on the screen they can be saved in an NC-program file. This NC-program is sent to an NC milling machine to produce your prototype.
Do note that two different versions of DeskProto are available: DeskProto **Full** and DeskProto **Lite**: the latter offering less functionality and available at a lower price.

File types
Three file types are supported to transfer the geometry from the CAD-system to DeskProto: STL, DXF and VRML. The first file type, the **STL-file**, is standard for all types of rapid prototyping, and defines the outer surface of the geometry using a large number of triangles. Any current 3D CAD-system is able to write a .STL-file, DeskProto can read this type both ASCII and binary. The other types, the **DXF-file** and the **VRML-file**, are supported only if they also contain a surface description in triangles.
In addition DeskProto can also import 2D files (DXF and EPS) for 2D Operations, and bitmap files (BMP, JPG and GIF) for Bitmap Operations. For 2D DeskProto can only process files containing simple 2D drawings.
All information on milling parameters, geometry, used machine and used cutters is saved in a DeskProto Project file (**DPJ-file**), available to quickly create a next version of the prototype.
The type of NC-program file that is written depends on the CNC milling machine that you have selected. DeskProto supports almost any CNC milling machine on the market.

1.3 Specifications

Required operating system:
Windows 2000 / XP / Vista or newer (Win 95 / 98 / ME / NT are no longer supported from V 5.0)

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

DeskProto is reported to run OK on an Apple Mac under Mac OS X using the Parallels Desktop Win XP emulation software. Even the USB dongle of the licensed version is recognized.

Required hard disk space:
About 10 MB for program only, plus at least 100 MB for projects

Required internal memory:
As much as possible, recommended at least 512 MB

Required connection:
A USB port is needed for the dongle (hardware key to prevent illegal copies).

Recommended:
A 3D graphics card that supports OpenGL

Supported Project files:
DPJ, version 1.1 (import only)
DPJ, version 2.0
DPJ, version 3.0 / 3.1
DPJ, version 4.0 / 4.1
DPJ, version 5.0

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

Supported Geometry filetypes:
STL STereoLithography
ASCII & binary

DXF AutoCAD Drawing Interchange File
limited to triangles and rectangles (3D)
VRML Virtual Reality Modeling Language
limited to triangles and rectangles

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

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

BMP Windows BitMaP
Used by the bitmap operation to load a bitmap
JPG Joint Photographic Experts Group
Used by the bitmap operation to load a bitmap
GIF Computerserve Graphic Interchange File
Used by the bitmap operation to load a bitmap

Supported NC program files (toolpath data):
NC-program Numeric Controlled
only ASCII, machine-dependent format and file extension

Maximum Filesize
The DeskProto code does not have a maximum for any file. However, because of the memory limitation for **32 bits** Windows versions (see above) the max filesize nevertheless is limited. As the memory needs to contain both the geometry and the toolpaths these are related: for a very large STL file (say 1 GB) only simple toolpaths will be possible. Roughly speaking the STL file size maximum for 32 bits DeskProto will be about 800 MB (binary STL).
The **64 bits** version can accept much larger STL files. This mainly depends on how much RAM memory is present: when Windows needs to swap the calculations will become too slow for most users.

1.4 Support

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

1. Search for a solution in the on-line **Help** system.

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

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

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

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

In case of any other problem: contact Delft Spline Systems.

1.5 Delft Spline Systems

Delft Spline Systems is a Dutch software house, founded in 1985, 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, and is 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 a module to easily calculate NC milling paths has been present from the start.

Since that time Rapid Prototyping has been recognized as vital for product development, has even become a buzz word. Based on a long experience of Rapid Prototyping and NC milling, Delft Spline Systems was able to develop this unique DeskProto package. We hope you will enjoy using it, and we expect it will help you in producing high quality designs within a short time.

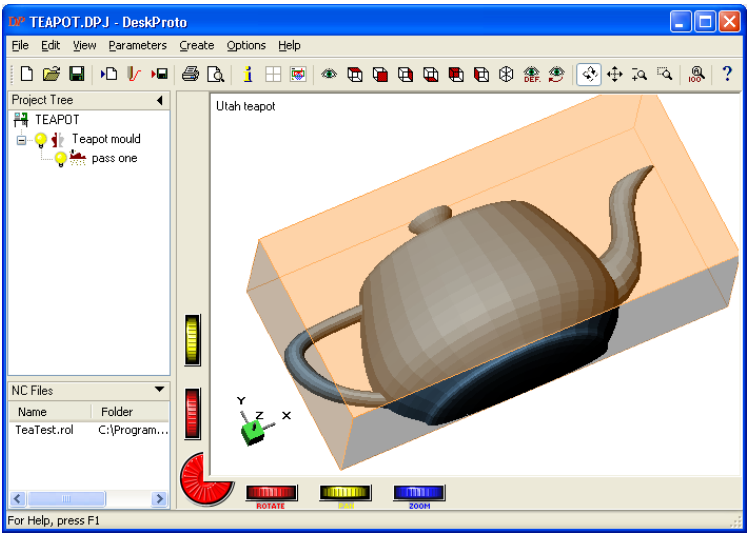
If you do want more information, register now; using the registration card included in the DeskProto box. You will be sent interesting information about new releases and other DeskProto-related subjects. It's a great opportunity to keep up with the latest developments in the promising world of computer aided design, and CNC-milling machines particularly.

Delft Spline Systems

PO.Box 2071
3500 GB Utrecht
The Netherlands

Tel:	**+31 30 296 5957**
Fax:	**+31 30 2962292**
Website:	**www.deskproto.com**
Email	**info@deskproto.com**

II Screen layout



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

- The top line is called the **Title bar**, default color blue. It contains the name of the current project, the word DeskProto and a few Windows icons.
- The **Menu bar** is the next horizontal line, default color grey. It contains in black characters the names of the available pull-down menus.
- The button bar or **Toolbar** is the horizontal line below (grey as well), containing a number of pictures (icons) that can be used as 'push-buttons'.
- The **View window**. The large screen-area below the button bar (right part of the screen) is used to display the geometry in one or more views.
- 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.

Note that you can **Resize** the screen, just as you can most Windows dialog

screens. Two standard sizes and Custom are available:

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

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

2.1 Title Bar

The Title bar is located along the top of a window. It contains the name of the application and the opened project. In the illustration above it says "Untitled" as the project has not yet been saved.

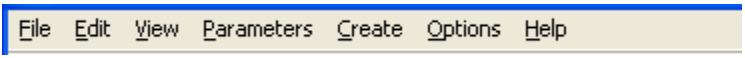
To move the window, drag the title bar. Note: You can also move dialog boxes by dragging their title bars.

A title bar may contain the following elements:

- Application Control-menu button (the orange icon "DP")
- Name of the project
- Name of the application
- Minimize button
- Maximize or Restore button
- Close button

The exact appearance will depend on your version of Windows, and on the Appearance settings that you have selected in Windows.

2.2 Menu Bar



The menu-bar is the main route to all the commands that are available in DeskProto. This is why the command reference in the next chapter of this manual is structured following the pull down menus. Each item in the menu-bar represents a pull down menu, which can be made visible by clicking the left mouse button with the cursor on the item. The following items 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 geometry and milling parameters.
Create	Toolpath calculation and saving options.
Options	Customization of libraries, defaults and other settings.
Help	Online Help options

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

2.3 Toolbar

The toolbar is the series of buttons displayed across the top of the application window, below the menu bar. The toolbar provides quick mouse 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 the menu item when the Toolbar is displayed.

The View menu also offers the command Large Toolbar buttons: very handy when you are working with a hight resolution screen.

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 geometry file into the project.

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

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

Print the image of all views.

Preview how the image would be printed.

Show or hide the geometry information dialog.

Change the Layout out of the views.

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


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


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


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


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


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


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


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


Set the viewpoint of the active view to Isometric view.


Set the viewpoint of the active view to Default view.


Restore the previous viewpoint settings.


Change mouse-function to rotation.


Change mouse-function to panning.


Change mouse-function to zooming.


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

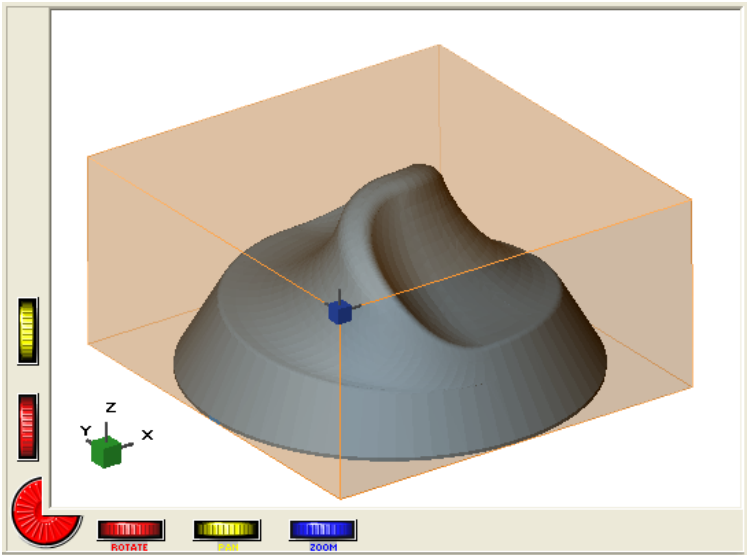
Note that one of these four mouse-functions is active at any moment, so choosing one means deselecting the previous one. DeskProto also offers other tools: the red thumb-wheels on the screen are for rotations, the yellow for pan and the blue wheel is for zoom. A handy alternative is using the middle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


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


Display the Help Topics dialog.


Enable Context sensitive help.

2.4 View Window



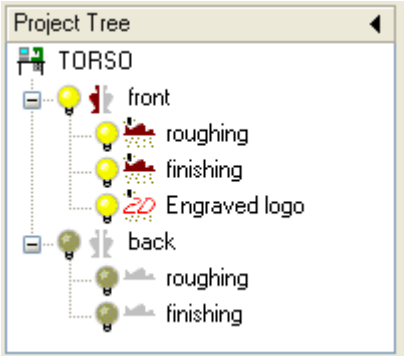
The view window shows the geometry that you are working with. You can have different representations of the geometry in your scene, for instance the original geometry drawn in lines, or the calculated toolpath. Which of these representations is shown can be defined with the Subjects in View dialog. Here you also can select other subjects, like title, orientator and working area.

In the drawing above the Segment has been made **Translucent**, making it more easy to recognize it as the block of material needed to machine this part. Translucency can be set in the Subjects in View dialog.

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. Do 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.
You can make the thumb-wheels visible and invisible using the

Thumb-wheels command in the View menu.

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, one with three operations and one with two. Two different Operation types are present: the (standard) Operation which is 3D, and the 2D Operation (note the different icon in the Tree).

The Lamp icon  indicates if a line in the Tree is visible (yellow) or not (grey). 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.

The lamp can also be red  , indicating an error status for that line.
Note that the icon also changes when you are editing the parameters for that Part or Operation:  indicates Edit status.

If you do not see a Project tree window, check the option Project tree in the View menu. A check-mark appears next to the menu item when the Project Tree is displayed. Un-checking this option 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 to open this Tree Window, and in the Project Tree window Title bar to close it.

The project tree offers you a number of functions:

Editing parameters

Double-clicking on an item 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 you should make it current. To make a part current just click with the left mouse-button on the grey 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 an a yellow lamp icon: instead you need to make a different part current.

Making an operation (in)visible
To be able to see how the toolpaths or z-grid looks 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 grey 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. The available functions will be different for each line of the tree, and will include the following options:

Edit the parameters of that tree-item.

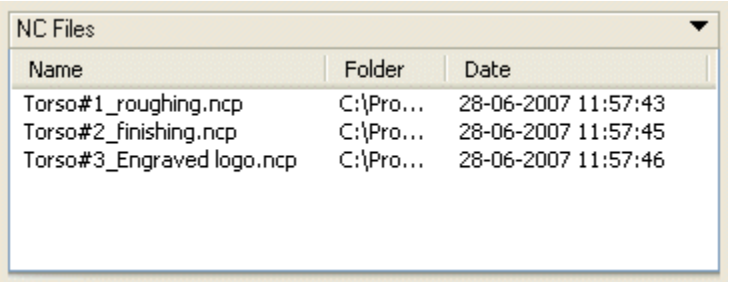
Add a Part to the project. The settings of the default part are used.
Add an Operation to that part. The settings of the first default operation are used. There are Add options for a 3D Operation, a 2D Operation and a Bitmap Operation.
Copy an operation or a part will add an item that is identical to the current one.

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

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

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

2.6 NC Files



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

By right-clicking on any line in this window you can open a Context menu, offering options to remove or delete the file, and also to send it to the machine (only in case that option has been configured).

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

2.7 Status Bar

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

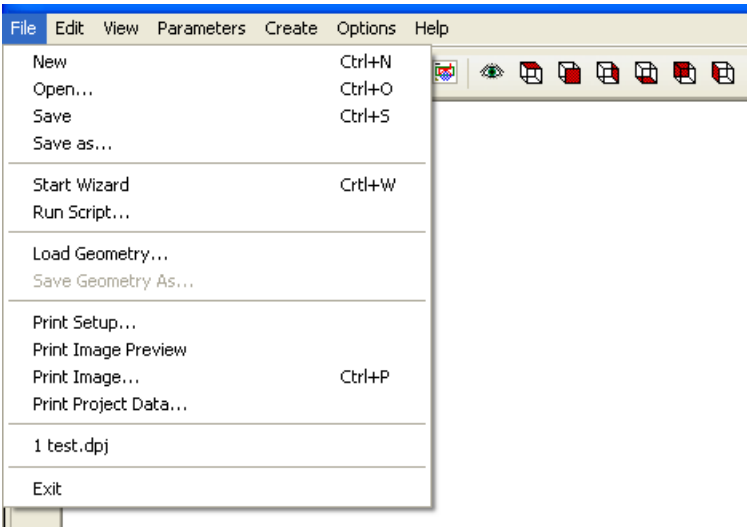
The left side of the status bar describes the actions of toolbar buttons as you point to them. Menu item actions are also described when you use the arrow keys to navigate through the menus.

In the middle area the most important parameters of the selected tree-item are displayed: geometry filename for project, machine and rotation for part, cutter and precision for Operation.

At he right side of the status bar the **coordinates of the current mouse position** are displayed, only when the geometry is displayed in one of the main views. These coordinate values shown are in “Transformed” coordinates: after the Scale, Mirror and Rotate transformations, but before Translation. This is a very handy option that enables you to quickly check dimensions and positions on the screen.

III Menu and commands

3.1 File Menu



The File menu contains all options for File management and for Printing, conform Windows conventions.

3.1.1 New command

This File menu command creates a new project in DeskProto. This project will be called "Untitled" until it has been saved.
As a next step you can then load a geometry.

When started DeskProto automatically starts a new project, so no need to again give this command.
You can open an existing project with the Open command.

Shortcuts:

Toolbar: 
Keys: CTRL+N

3.1.2 Open command

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 DeskProto project files (.DPJ).

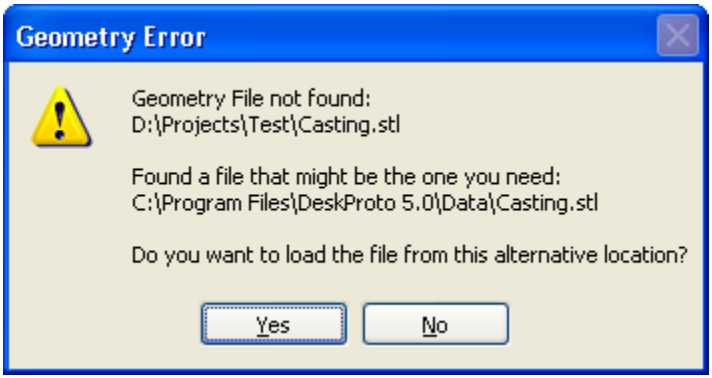
You can't open more than one project at a time.
You can create a new project with the New command.
A 'Close' option is not present: you can close your current project using either 'New', 'Open' or 'Exit'.

Note that you can NOT load a geometry file using the Open command: only project files that you have already worked on with DeskProto. To open a geometry file you need to create a New project and use the command Load Geometry.

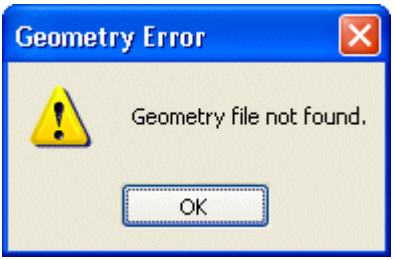
Shortcuts:

Toolbar: 
Keys: CTRL+O

When opening a project file, the geometry file that was used will be read again: the DPJ files does not store the geometry but only a link to the Geometry file. The geometry file must thus be found at the same place as where it was when the project file was saved. If the geometry file cannot be found (for instance when the DPJ file was copied from a different computer system) then DeskProto will check if a geometry file of 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:



If the file also cannot be found on any alternative location, then this error-message will be displayed:



After pressing the OK button the Project Parameters dialog will appear, giving you the opportunity to select the correct geometry file.

3.1.3 Save command

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 displays the Save As dialog so you can name your project.

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

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

Shortcuts:

Toolbar: 
Keys: CTRL+S

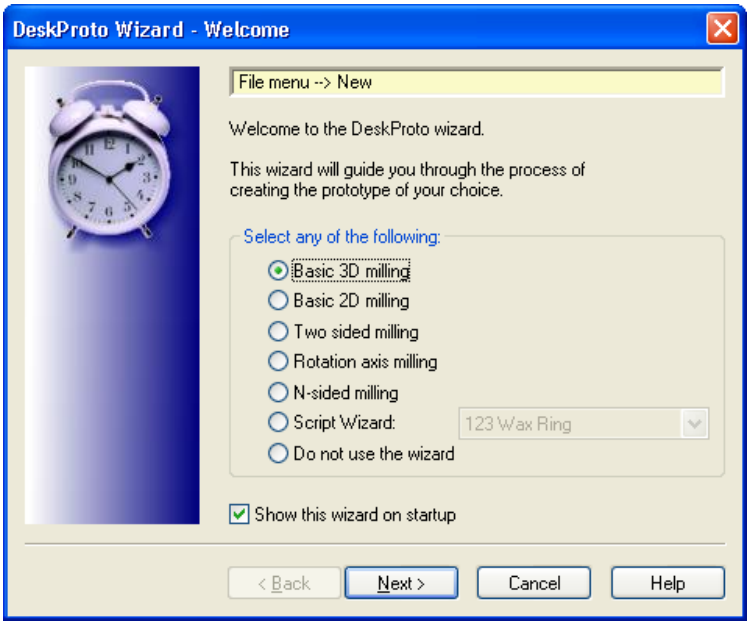
3.1.4 Save As command

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: with or without Toolpaths.

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

3.1.5 Start Wizard command



This File menu command presents you with seven possible choices, of which you have to select one before continuing .

Basic 3D Milling
This is the “Single Sided Milling Wizard”, which shows the simplest way to machine a model from your geometry. It is meant for novel DeskProto users, and explains step-by-step the procedure to create an NC toolpath file (NC program) based on your geometry. The model will only be machined from one side, hence the name of this wizard.

Basic 2D Milling
This wizard shows how to create 2D toolpaths for just one 2D DXF file. It is meant for novel DeskProto users, and explains this procedure step-by-step. As DeskProto is in fact meant for 3D applications, 2D milling is more complicated than 3D (it is in fact hidden). This wizard explains where to find it.

Two Sided Milling (Not available in the Lite version)
The "Double Sided Milling Wizard". This advanced wizard is a unique feature of DeskProto, and makes it very easy for you to create a compete 3D model by machining it from two sides. DeskProto assists you by taking care of the repositioning needed to machine the second half: no need to change the workpiece zero point (start position of the cutter).

Rotation Axis Milling (Not available in the Lite version)
If your machine is equipped with a rotation-axis (A-axis) you can use this wizard to create the toolpaths for a model that is machined from all sides. The rotation axis is an extra piece of equipment that lets your model rotate during machining (just like the meat rotating on a spit above a barbecue).

N-Sided Milling (Not available in the Lite version)
N-sided milling is also meant for machines with a rotation axis. Instead of rotating during machining as in the previous wizard, this wizard generates a toolpath to machine the part from several (N) sides, with just one positioning rotation in-between. This setup makes it possible to also machine areas that would be an undercut for rotation axis machining.

Script Wizard
This option gives access to a completely different class of Wizards. The wizards mentioned above are part of DeskProto, script wizards are separate files that can later be added. Any user or reseller can add script wizards to DeskProto, for more information see the Script Wizard page. When choosing this option the combo box on this line can be accessed by which you have to choose which script wizard to start.

Do not use the Wizard
When choosing not to use the wizard using this option, pressing the Next button will result in stopping the wizard (so this is in fact the same as just pressing Cancel).

Further see the Help screen for each Wizard page.

Do note that 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. The yellow line on top of each wizard page shows where to find the current parameter in the 'normal' interface. Also, after finishing any wizard you can still fine-tune the settings that the wizard made.

The checkbox **Show this wizard on Startup** makes the automatically appear at each start of DeskProto. After having deselected this option, you can still access the wizard using the command "Start Wizard" in the File menu. When changing the state of this checkbox make sure to "Finish" the wizard (not press "Cancel").

3.1.5.1 Script Wizard pages

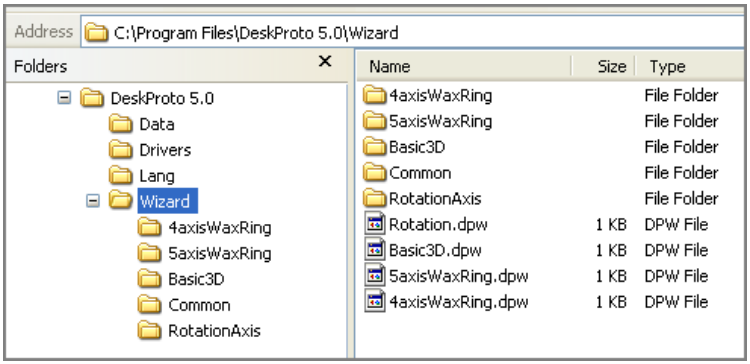
The Script Wizard is a wizard written in a Script language and later added to DeskProto.
This is a very powerful option, as it makes it possible for any user and reseller to create a custom wizard for a specific application and/or a specific fixture.

Using a Script Wizard.

Script Wizards can be accessed via the DeskProto Wizard Welcome page, where an combo box is present to select which Script wizard to use. This combo box is filled when DeskProto starts, with all valid Wizards found in the subdirectory \Wizard\ of the DeskProto directory, for instance in C:\Program Files\DeskProto 5.0\Wizard\

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

Creating a Script Wizard.



A valid Script Wizard consist of one DPW file (DeskProto Wizard) and a subdirectory with the same name.
For instance the file Basic3D.dpw and the directory C:\Program Files \DeskProto 5.0\Wizard\Basic3D\
The dpw file specifies the name of the Wizard, how many pages are present, and the files that define each page. These files typically are present in the subdirectory just mentioned.

Each wizard page is defined by an HTML file that also contains a script. The

HTML part defines the user interface (dialog design, texts, edit boxes, etc), and the script defines the actions to be performed. As scripting language both JScript and VBScript can be used.

So basically all you need to add a Script wizard to DeskProto is a plain text editor. Copy the resulting files to the location specified above, and DeskProto will show your new wizard.

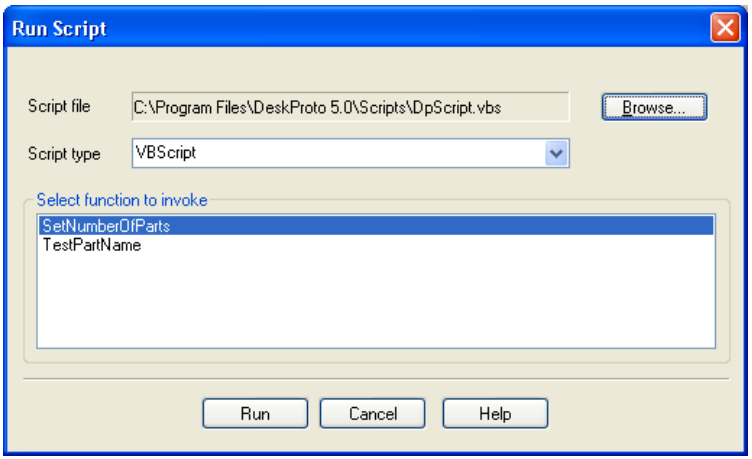
For more information on scripting see the Script page, and the DeskProto Script Reference on the DeskProto distribution CD. Note that a very easy way to create a Script wizard is to copy and rename one of the sample Script wizards: then the correct structure already is present.

3.1.7 Run Script

Normally when you use DeskProto you will load a geometry, set a number of parameters, calculate toolpaths and save an NC program file. So a number of tasks that are manually done or started in a specific sequence. In some applications the tasks 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 tasks to be done, saved in a file. These tasks of course need to be described in a language that the computer will understand: in a DeskProto script you can use both the JScript and the VBScript scripting languages. For more information also see the Scripts page.

You can start a script via the "Run Script..." command the File menu.



This command will open the Run Script dialog as shown above. Here you can browse the Script file: files are seen with extensions .js for JavaScript and .vbs for VBScript.
The Script file defines one or more functions: you need to select which function to invoke, and then press the Run button to start it.

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

Note that a special type of Script does exist, called the Script Wizard, which does exactly as the name suggests.

```
Function SetNumberOfParts()
    Dim nParts

    nParts = CInt(InputBox("How many parts would you like to have?"))

    If nParts > 0 Then
        DeskProto.project.numberOfParts = nParts
    End If
End Function

Function TestPartName
    DeskProto.project.activePart.name = "A New Part Name"
End Function
```

Above you see the very simple script file that was opened in the dialog just shown. It contains two functions, one to automatically create a number of parts, and one to rename the current part.

For more information on scripting see the DeskProto Script Reference on the DeskProto distribution CD.

3.1.8 Load Geometry command

The File menu command **Load Geometry** displays the standard Windows Open File dialog in which you can select the geometry-file you want to load. It will be showing Geometry files that DeskProto can read:
STL: STereoLithography File
DXF: AutoCAD Drawing Interchange File
VRML: Virtual Reality Modeling Language
After browsing the file it's contents will be loaded.

The Load Geometry command can only be used for 3D Geometry files of the above types. To load a 2D plotfile you need to open it in the 2D Operation

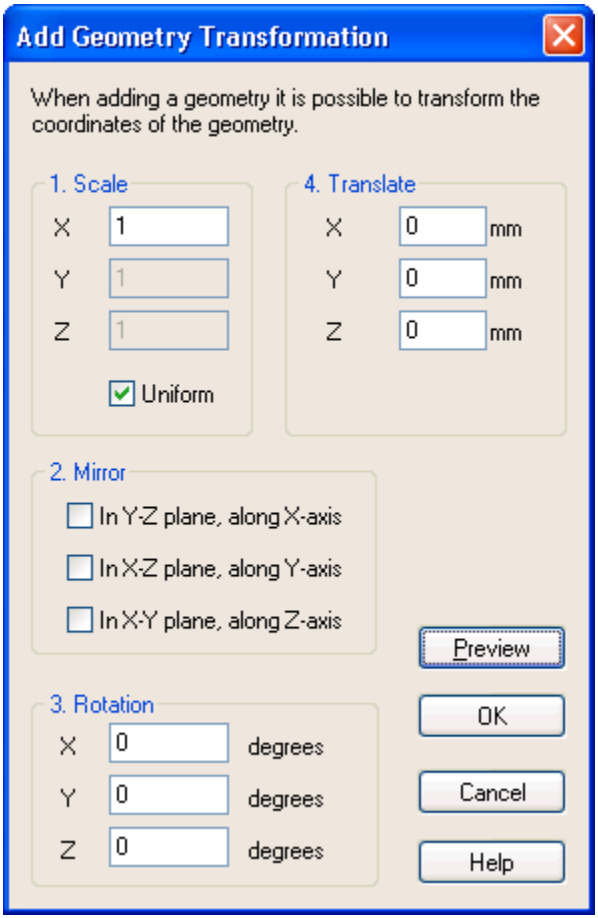
Parameters, to load a Bitmap file you need to open it in the Bitmap Operation parameters.

Shortcuts:

Toolbar: 

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

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



It offers basically the same transformations as also present in the Part Parameters (see there for more help), however these will be applied to the new geometry file only. After adding the second geometry file DeskProto only sees one resulting geometry, making it impossible to later translate or rotate only the new geometry elements.

A very handy tool in this dialog is the **Preview** button, as it will draw the new geometry file according to the transformations that you entered in the dialog. If this new position is not OK, you can enter new transformation values and again press Preview for a new drawing. When all is OK you can acknowledge with OK, or abort with Cancel.
Note that on top of the Transformations that you enter in this dialog, also the Part Transformations of the visible part will be applied to the drawing. This

means that when a rotation is applied in the Part Parameters setting a correct Rotation for Add geometry will be very difficult.

An application for this extra transformation is for instance if you want to machine two parts in one setup, positioned next to one another.

3.1.9 Save Geometry as command

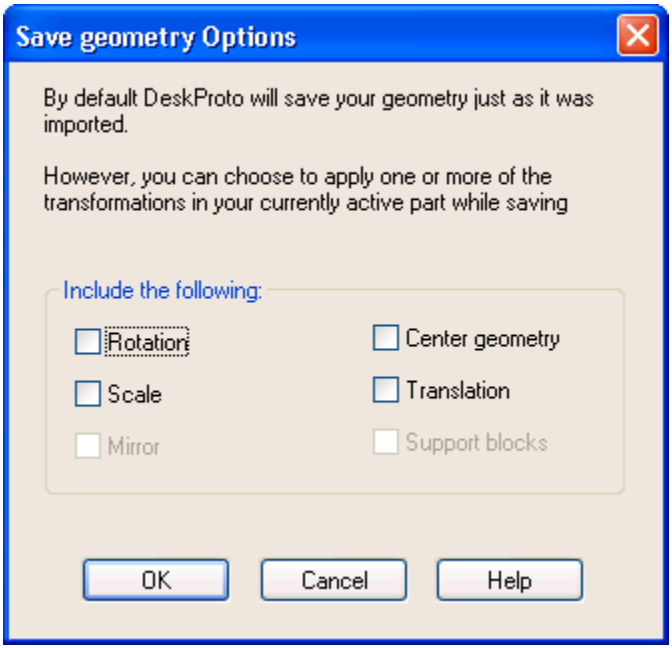
This File menu command displays the standard Windows Save File dialog in which you can define the geometry-file you want to write. This command makes it possible to use DeskProto as a converter for geometry files. It also can be used to merge several geometry files to a combined new file.

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

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

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

Before this standard Save dialog an extra dialog will pop up: the **Save Geometry Options** dialog, which enables you to apply Transformations that you have set in the Part Parameters. The dialog looks like this:



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 than contain rotated geometry, scaled geometry, with support blocks, etc. This is ideal for instance for saving a scaled or rotated version of your geometry file. This feature also can be used to create more than 4 support blocks.

3.1.10 Print Setup command

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

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

3.1.11 Print Image Preview command

This File menu command displays the image of the View Window as it would appear when printed.

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

Shortcuts:

Toolbar: 

3.1.12 Print Image command

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

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

Shortcuts:

Toolbar: 
Keys: CTRL+P

3.1.13 Print Project Data command

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

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

3.1.14 File 1, 2, 3, 4 command

Use the numbers and filenames listed at the bottom of the File menu to open one of the last eight projects you closed. Choose the number that corresponds with the project you want to open.

This list is called the **Recent File List**, and is a convenient standard option of current Windows software.

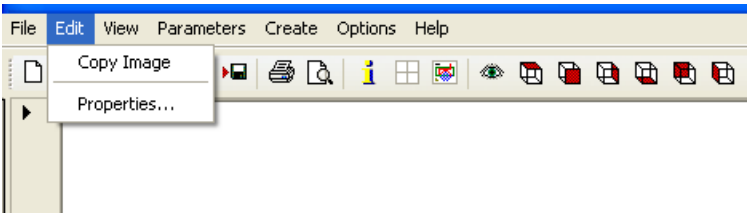
3.1.15 Exit command

This FIle menu command ends your DeskProto session. You can also use the Close command on the application Control menu. DeskProto prompts you to save projects with unsaved changes.

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

Keys: ALT+F4

3.2 Edit Menu



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

3.2.1 Copy Image command

This Edit menu command copies the contents of the DeskProto View Window to the Windows Clipboard.
The clipboard is used to cut and paste data between windows applications: you can for instance Paste this image into a Word document.

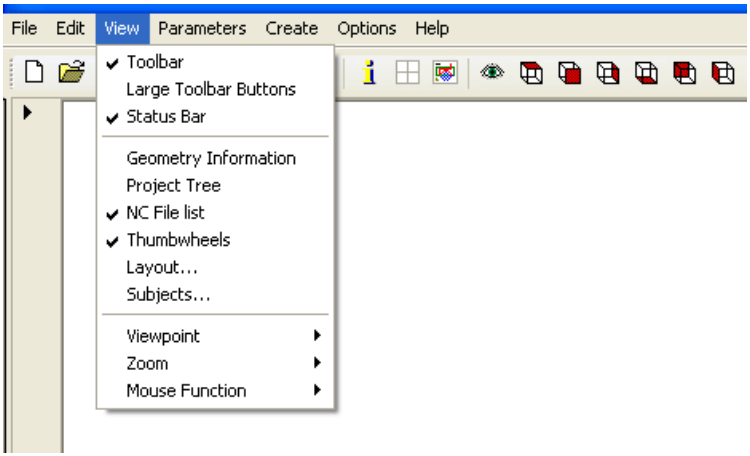
Copying data to the clipboard replaces the contents previously stored there. Note that this command has no visible effect in DeskProto.

Shortcuts:
Keys: CTRL+C

3.2.2 Properties command

This Edit menu command can be used to edit the highlighted tree-item or the active view. Depending on what is selected last this command will open one of the following five dialogs:
Project Parameters , Part Parameters , Operation Parameters , 2D 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.

3.3 View Menu



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

3.3.1 Toolbar

The toolbar is the series of buttons displayed across the top of the application window, below the menu bar. The toolbar provides quick mouse 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 the menu item when the Toolbar is displayed.

The View menu also offers the command Large Toolbar buttons: very handy when you are working with a hight resolution screen.

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 geometry file into the project.


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


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


Print the image of all views.


Preview how the image would be printed.


Show or hide the geometry information dialog.


Change the Layout out of the views.


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


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


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


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


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


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


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


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


Set the viewpoint of the active view to Isometric view.

Set the viewpoint of the active view to Default view.


Restore the previous viewpoint settings.


Change mouse-function to rotation.


Change mouse-function to panning.


Change mouse-function to zooming.


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

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


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


Display the Help Topics dialog.


Enable Context sensitive help.

3.3.2 Large toolbar buttons

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

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

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

3.3.3 Status bar

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

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

In the middle area the most important parameters of the selected tree-item are displayed: geometry filename for project, machine and rotation for part, cutter and precision for Operation.

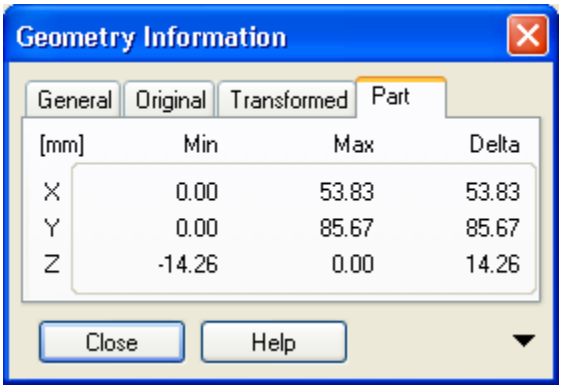
At he right side of the status bar the **coordinates of the current mouse position** are displayed, only when the geometry is displayed in one of the main views. These coordinate values shown are in “Transformed” coordinates: after the Scale, Mirror and Rotate transformations, but before Translation. This is a very handy option that enables you to quickly check dimensions and positions on the screen.

3.3.4 Geometry information dialog

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

Shortcuts

Toolbar: 



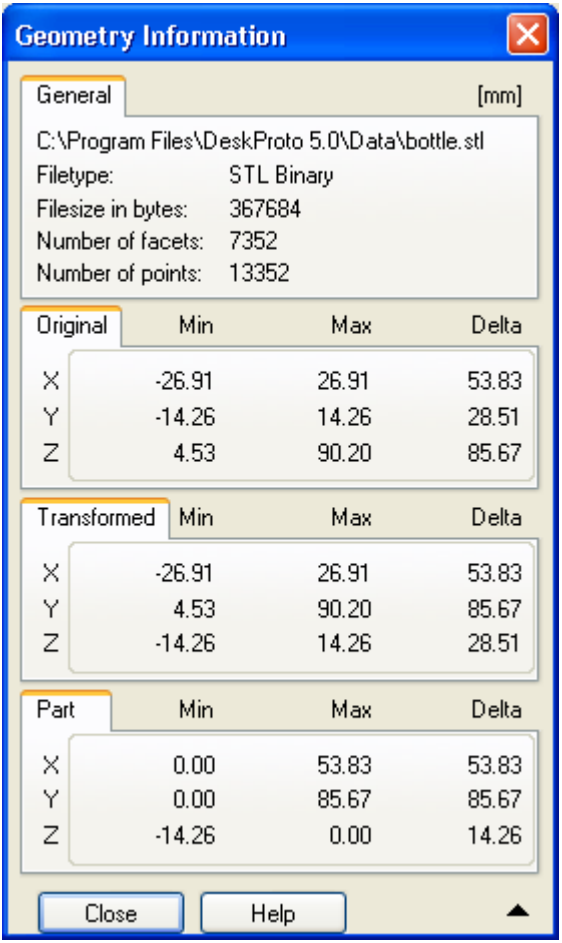
Depending on which Operations are present in the project, four, five or six tab pages are shown.

General:
Information about the file: file name, location, size;
Information about its contents: number of facets (triangles) and points.
Original:
The dimensions of the original geometry, as read from the geometry file (CAD-coordinate system). These values are the same for all parts.
Transformed:
The 'transformed' values are an intermediate result (for the current part): all part parameters from the Transform page have been applied (scale, mirror and rotate). The values are in an ‘in-between coordinate system”.
Part:
The dimensions of the prototype as it will be machined, so after segmentation and translation have been applied as well. These are given in the workpiece coordinate system set on your milling machine. This is important information for you, as these are also the dimensions of the Block of material that you have to prepare.
2D (in case present):
The dimensions of the 2D Operations (only the visible Operations), in the workpiece coordinate system.
Bitmap (in case present):
The dimensions of the Bitmap Operations (only the visible Operations), in the workpiece coordinate system.

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 can be used to prepare the block of material to be machined.

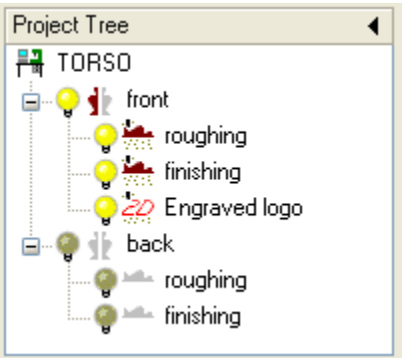
The dialog can be viewed in two ways; as the small dialog box with tab

pages (shown above), or as a large dialog box with all the information visible at the same time (shown below). You can switch easily between those two types of dialogs by pressing the button at the bottom on the right side (the small black arrow).



The dialog can stay open while you are working with the project.

3.3.5 Project Tree



In DeskProto the **structure** of the Project is shown in the project tree, which is placed on the left of the screen. In the above example the project 'Torso' contains two parts, one with three operations and one with two. Two different Operation types are present: the (standard) Operation which is 3D, and the 2D Operation (note the different icon in the Tree).

The Lamp icon indicates if a line in the Tree is visible (yellow) or not (grey). 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.

The lamp can also be red , indicating an error status for that line.
Note that the icon also changes when you are editing the parameters for that Part or Operation: indicates Edit status.

If you do not see a Project tree window, check the option Project tree in the View menu. A check-mark appears next to the menu item when the Project Tree is displayed. Un-checking this option 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 to open this Tree Window, and in the Project Tree window Title bar to close it.

The project tree offers you a number of functions:

Editing parameters

Double-clicking on an item 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 you should make it current. To make a part current just click with the left mouse-button on the grey 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 an a yellow lamp icon: instead you need to make a different part current.

Making an operation (in)visible
To be able to see how the toolpaths or z-grid looks 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 grey 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. The available functions will be different for each line of the tree, and will include the following options:

Edit the parameters of that tree-item.

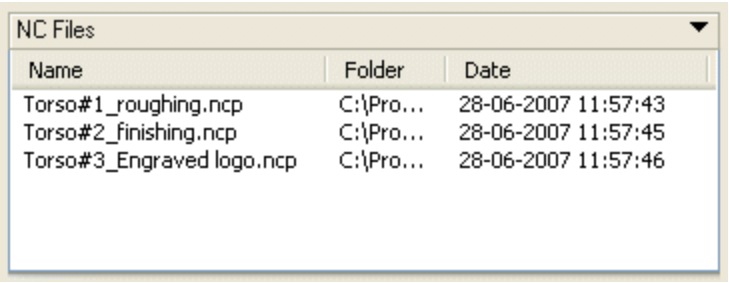
Add a Part to the project. The settings of the default part are used.
Add an Operation to that part. The settings of the first default operation are used. There are Add options for a 3D Operation, a 2D Operation and a Bitmap Operation.
Copy an operation or a part will add an item that is identical to the current one.

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

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

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

3.3.6 NC Files list

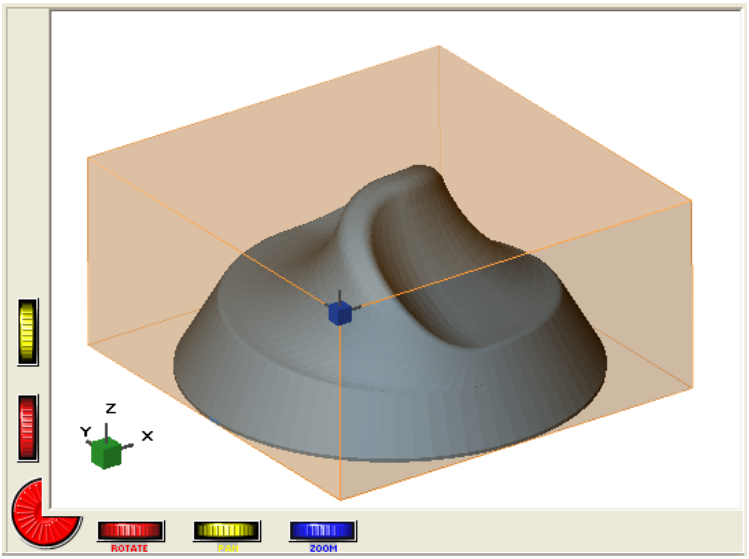


The NC files window shows a list of NC program files that have been saved for this project. For each file name, location and time are listed. This may be easy for you to manage NC files for this project.

By right-clicking on any line in this window you can open a Context menu, offering options to remove or delete the file, and also to send it to the machine (only in case that option has been configured).

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

3.3.7 Thumbwheels



The view window shows the geometry that you are working with. You can have different representations of the geometry in your scene, for instance the original geometry drawn in lines, or the calculated toolpath. Which of these representations is shown can be defined with the Subjects in View dialog. Here you also can select other subjects, like title, orientator and working area.

In the drawing above the Segment has been made **Translucent**, making it more easy to recognize it as the block of material needed to machine this part. Translucency can be set in the Subjects in View dialog.

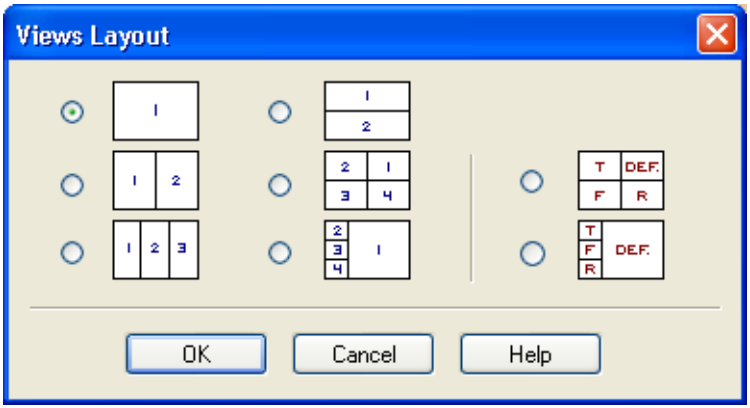
The **Thumb-wheels** that are drawn in the border of the view window offer an easy way to change the camera position. You can use them by pressing the left mouse button with the cursor on the wheel and then moving the mouse, keeping the left button pressed. The cursor will become arrow-shaped to guide you. Do 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.
You can make the thumb-wheels visible and invisible using the Thumb-wheels command in the View menu.

3.3.8 Layout dialog

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: 



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 Subjects in view. 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 geometry looks like.

Note:
In a Layout with more than one view, one of the views will be the current (active) view. You can make a view current by clicking your left mouse-button inside the view: do observe the blue line that appears around the active view. The thumb-wheels, view buttons and view commands apply only on the current view.

3.3.9 Subjects dialog

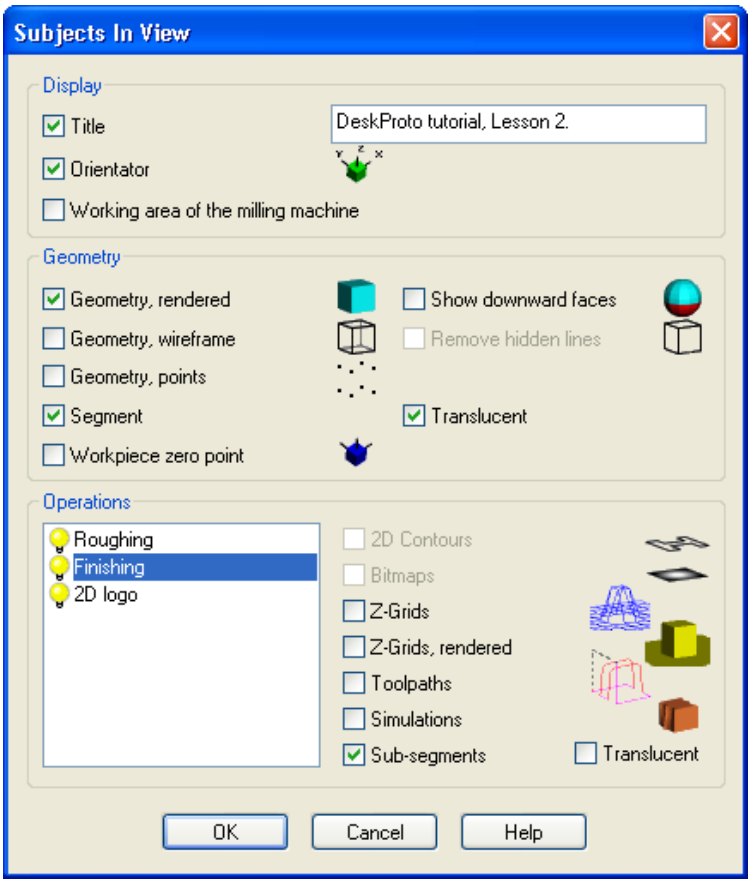
The View menu command Subjects... displays the **Subjects in View dialog,** in which you can change what is and what is not shown in the active view (the Scene). In case none of the boxes is checked the View Window will be blank. You can also select which operations should be visible.

Shortcuts

Toolbar: 

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

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



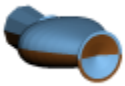
The **Title** will be displayed on the screen and on the printed view, in the top left corner of the View.

The **Orientator** is the coordinate system icon (cube with axes) displayed in the bottom left corner of each view. It helps you to understand from which direction you are looking at the geometry. This green cube does NOT indicate the zero point.

The **Working Area** can be drawn, which is the working area of the machine selected for this part. Of course DeskProto does not know where you will set the workpiece zero point, so it will draw the part exactly in the middle of the working area. This will give a good indication how the part relates to the machine. The working area for your machine can be set in the Machine dialog.

The second group shows the Geometry-related subjects:

In a **Rendered geometry** drawing all triangles of the geometry definition are made "solid" with color. This is the default Subject in View as this offers a good understanding of your geometry.

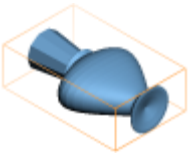


The sub-option **Show downward faces** will assign a different color to any triangles that are facing down (of which the normal has a negative Z-component). This option makes it very easy to check for Undercuts, and to optimally rotate your geometry to reduce undercuts.

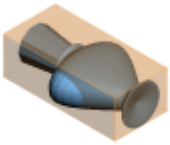
In a **Wireframe geometry** the triangles are drawn with lines.
it is possible that you can't see the geometry very well because all the lines are confusing.

In such case the 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. Of course these are only visible in case Wireframe geometry has not been not selected.



The **Segment** shows the rectangular bounding box of the part in light brown lines. This is in fact the block of material that is needed to machine your part. Note that often only the green lines of the sub-segment will be visible, when both segments are equal.



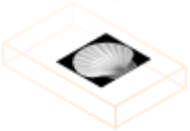
The sub-option **Translucent** draws the sides of the segment in a translucent color (for the Part segment in brown). This will shown the segment block more clearly, maintaining visibility of the geometry inside.

A second **Orientator** is available at request, drawn exactly at the workpiece zero point (in blue). The position of the Workpiece zero point can be changed using the Translation options of the Part Parameters. Note that the small black dot on the screen is NOT the zero point: it indicates the first point of the toolpath.

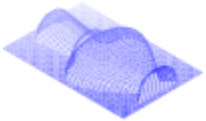
The selection of subjects in the third group may be different per Operation. It is for instance possible to show the toolpaths of Operation Roughing and not show them for Operation finishing, while both operations are visible.

In the list of **Operations** you can change the visibility of each Operation by clicking on the lamp icons. Yellow (lamp on) means visible, gray (lamp off) means invisible. A red lamp indicates an error status for that Operation. This is an alternative for clicking the lamp icons in the Project Tree.
In the list you can also *Select* one or more Operations: making the line blue (meaning selected) by clicking on it. Two or more operations can be selected by keeping the Control or the Shift button depressed when clicking. The Subjects that you will check and uncheck for the Operations, apply ONLY to the selected operations.

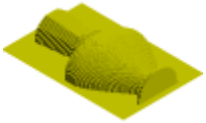
2D Contours concern the 2D drawing(s) that are used in 2D Operations. This option is available only when one or more 2D Operations are selected.



Bitmaps concern the Bitmap drawing(s) that are used in Bitmap Operations. This option is available only when one or more Bitmap Operations are selected.



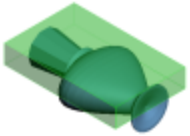
When you check the **Z-Grids** checkbox the Z-grids of the visible operations will be shown. The Z-grid is an intermediate representation of the geometry that DeskProto uses for it's toolpath calculations. The Z-grid will be drawn in lines.



The same subject will be show for option **Rendered Z-Grids**. Now the Z-grid is drawn as a rendering, so you can clearly see that it is a 3D bar graph representation of the geometry.

When you check the **Toolpaths** checkbox the Toolpaths of the visible operations will be shown. The toolpath that is drawn is the same toolpath that will be sent to the machine: if there are any problems it should be possible to detect them now.

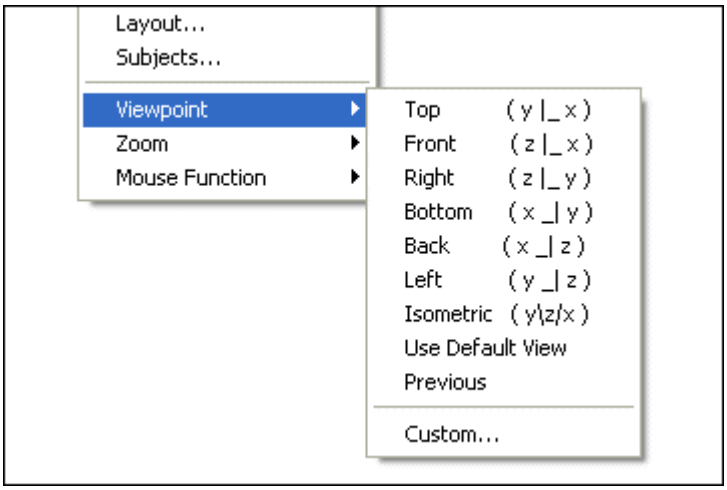
When you check the **Simulations** checkbox the Simulations of the visible operations will be shown. You can clearly see what the resulting model will look like.



When checked a **Sub-segment** will be shown: the rectangular bounding box of the operation in light green lines. The sub-option **Translucent** draws the sides of the segment in a translucent green color. This will shown the segment block more clearly, maintaining visibility of the geometry inside.

The colors that are used for the various subjects in the example drawings above are the DeskProto default colors. These can be changed in the Colors tab of the DeskProto Preferences.
For all line drawings default fog may be applied to simulate depth, which can be set on the same location.

3.3.10 Viewpoint submenu commands



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

Top / Front / Right / Bottom / Back / Left to set one of the six main views.
Isometric / Use default to set the Isometric or the Default view.
Previous to return to the previous View settings.
Custom will open the Viewpoint dialog.

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

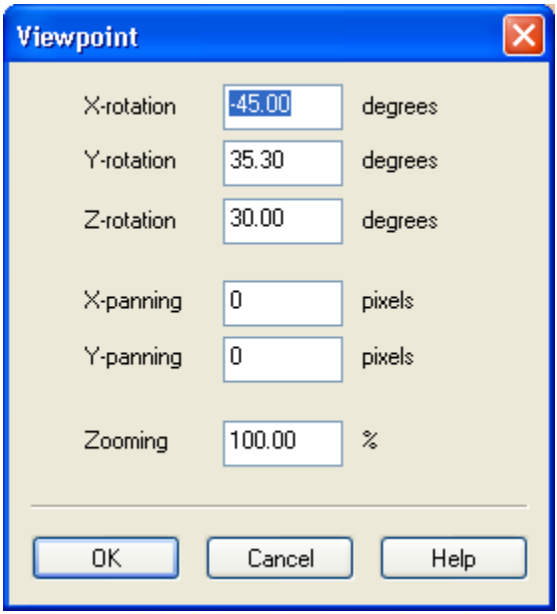
3.3.11 Viewpoint dialog

The View menu command Viewpoint ->Custom... displays the **Viewpoint dialog**, in which you can change the settings of the active view, that is the rotation, panning and zooming settings. In fact this is setting the Camera position.

Shortcuts:

Toolbar: 

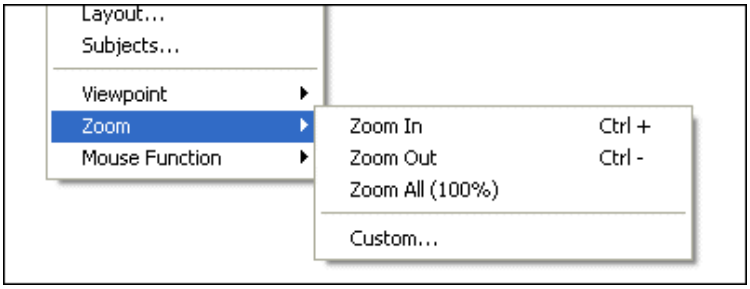
Mouse: Right-click in a view, and select the Viewpoint item in the shown context-menu.



In this dialog you can change the point of view from which you look at the geometry, z-grid, toolpath etc. You can set the values exactly, using the keyboard. The rotations are executed in the order X, Y, Z. The effect is the same as when using mouse rotation, pan and zoom or when using the thumbwheels.

Note:
These values do not change the way the geometry will be milled (only the Camera position). To change those settings you should go to the Transform tab page of the Part Parameters dialog.

3.3.12 Zoom submenu commands

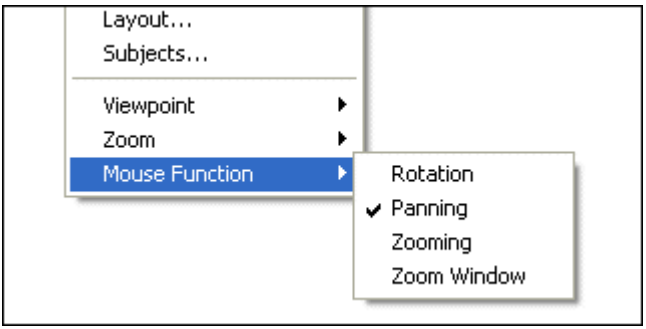


The View -> Zoom submenu offers the following commands:

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

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

3.3.13 Mouse Function submenu commands



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

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

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

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

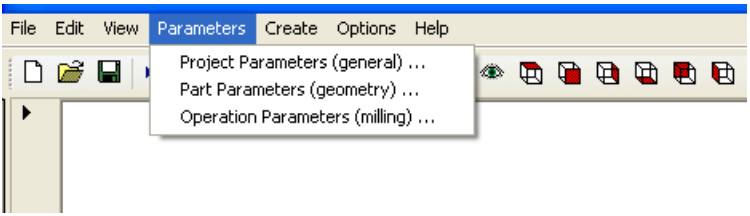
Zoom Window: use the mouse to zoom into any part of the screen. Click the left mouse button to define one corner of a bounding box, move the mouse keeping the button pressed, and release it as you have reached the opposite corner. The part of the screen inside the bounding box will now be displayed as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

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

An alternative for *Zoom* is rotating the **Mouse wheel**. This works no matter which of the above buttons is active. 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 button.

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

3.4 Parameters Menu



The parameters menu offers access to all milling parameters. The three main levels Project, Part and Operation follow the structure of the Project Tree: one Project can contain one or more Parts (for instance a left half and a right half), and each Part can contain one or more Operations (for instance for roughing, finishing and detailing). Operations can be one of three types: the standard 3D Operation, the 2D Operation and the Bitmap Operation.

3.4.1 Project parameters command

This Parameters menu command displays Project Parameters dialog, in which you can edit the parameters of the project.

Shortcuts:
Double-click on the project-item in the project tree (the first line).
Or right-click on this line and select Project Parameters in the context-menu.

3.4.2 Project Parameters dialog

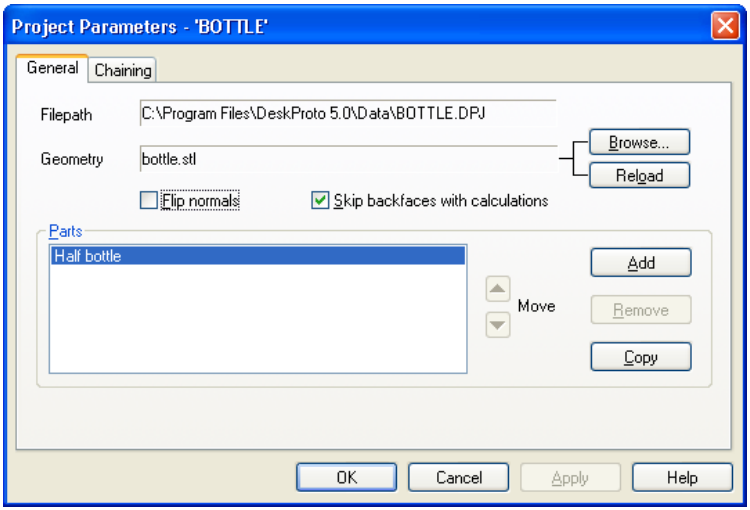
The Project parameters are divided into 2 sections by Tab pages, both of which will be described below.

Do 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, first option.
Or you can double-click on the project-item in the project tree (the base level item).
Or right-click on the project-item and select Project Parameters in the context-menu.

This same dialog is used for the Default Project parameters.

General parameters



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

Geometry
The geometry field contains the path and the name of the Geometry file being used in this project. You can change the geometry by pressing the **Browse** button at the right of the geometry field. When changing the geometry within an existing project, all Parts and Operations will remain the same; they will not automatically be reset or adjusted to the new geometry. For starting with new geometry it is therefore better to start a new project. The button **Reload** can be used to again import the file if it has been changed.

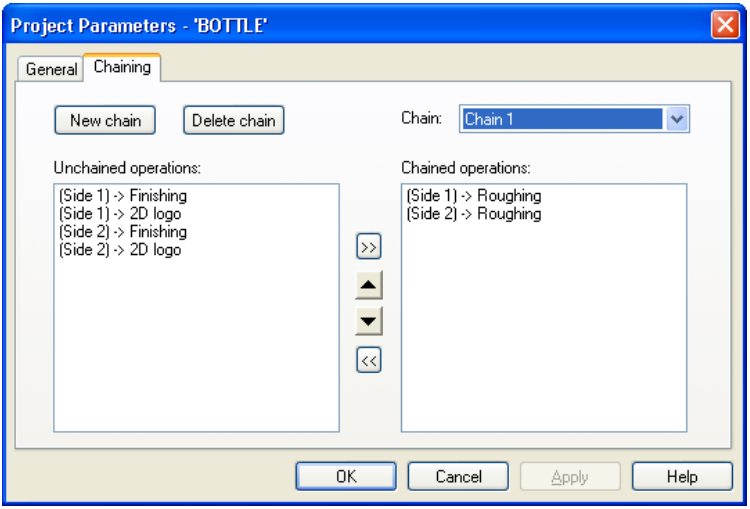
Flip Normals
In DeskProto for each facet (triangle) a normal vector is stored, indicating which side of the facet is on the outside of the geometry. This information is stored in the STL file. When using a corrupt STL file this information can be wrong. In case ALL normal-vectors point to the inside of the geometry you can use this option Flip normals to correct the normal information.

Skip Backfaces

For heavy calculations you may want to check the option Skip Backfaces in order to save calculation time. This option will skip all facets (triangles) of which the invisible inside (the backface) is on top as seen from the positive Z-direction. This means that half of the triangles can be skipped, which results in shorter calculation time. The resulting toolpath is the same (for a valid geometry). As the difference in calculation time is not very large the option is no longer checked by default.
For corrupt geometries you may need to disable this function.
This option will also influence the rendered drawing of the geometry on screen: when checked all backfaces will be drawn in black, when not both sides of each facet will be rendered..

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

Chaining parameters



Chaining is an option meant to combine Operations in different Parts into

one large NC Program file. Operations in the same Part will be combined into one NC Program file anyway. This option is used by the N-side milling wizard.
Note that chaining is of course only available in case more than one Operation is present in the current project.
Also note that Chaining is an option for advanced users.

Chaining can be achieved on two different locations in DeskProto:
- in the Start/End commands of the Operation parameters you can chain the current operation to be followed by a next one. So here you can only see one element of one Chain.
- this Chaining tab of the Project parameters gives you an overview of all Chains in this project, and allows you to edit them.

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

The button **Delete Chain** 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.

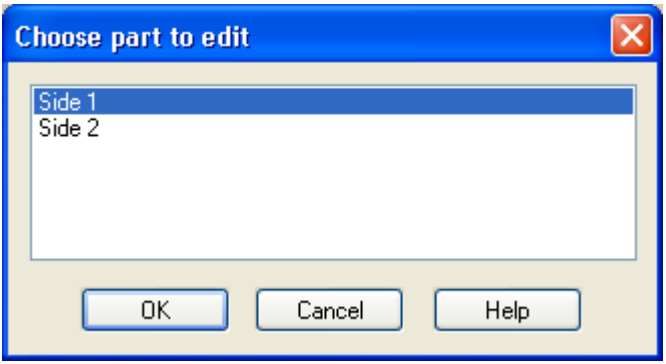
The field **Unchained Operations** shows all operations in the project that are not (yet) part of a chain. You can use the **left and right arrow buttons** to move unchained operations to a chain and to remove operations from the selected chain.

Note that in the Start/End commands you can also add extra commands to be executed in-between the two chained operations: either after the first Operation or before the subsequent operation.

3.4.3 Part parameters command

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

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



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

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

3.4.4 Part Parameters dialog

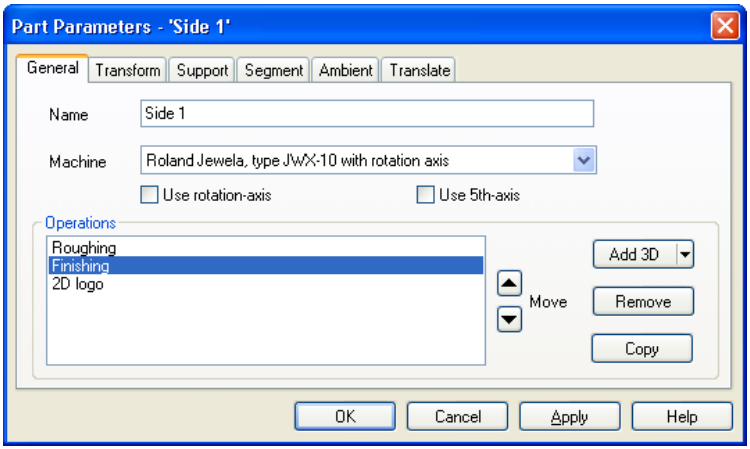
The Part parameters are divided into 6 sections by Tab pages. The parameters are applied to the geometry in the sequence as presented by the Tab pages, from left to right. Also: the further to the right, the more advanced the parameters in the tab. In DeskProto Lite only the first two Tab pages are available, as it does offer less parameters than DeskProto Full. The second Tab page also is different for Lite and Full.

Do also note that 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 Geometry information dialog.

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

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

General parameters



Name
The name of the part can be changed here. Use a meaningful name to easily remember the purpose of each specific part: the Project Tree then will be easy to interpret in case of more than one part. This name is for your convenience only: it is not used in the resulting NC program file.

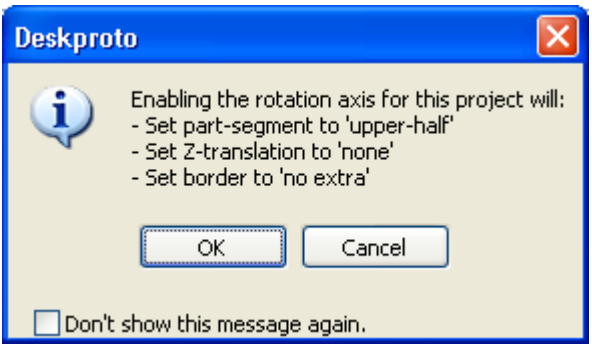
Machine
This refers to the machine you want to use. DeskProto uses the machine information to check whether the prototype is not too large for the machine, and to check whether the milling parameters (speeds) that you will enter are possible for this machine. The choice of machine also defines the format of the NC Program file, because the machine information states which postprocessor (driver) to use. You can have a look at the machine-definition using the option Library of machines in the Options menu.

Normally you do not have to bother with this parameter as your machine will be the default machine. If not, you can change the default machine using the option Default Part Parameters in the Options menu.
Note: when you change the selected machine DeskProto will if needed also change values for Feedrate and Spindle speed, as the new machine may not be capable of outputting the old values.

Use rotation-axis
A rotation axis is a device on the machine that rotates the part during machining. Like a piece of meat above the barbecue. DeskProto supports a rotation axis parallel to the X-axis, which is officially known as an A-axis. Note that the difference between an A-axis and a lathe: on a lathe the rotation of the Part causes the cutting, on a rotation axis the rotation of the

tool.DeskProto supports continuous rotation during machining.

This option is only available in case a machine has been selected that supports a rotation axis, as defined in the Library of machines This option is not available in DeskProto Lite. If this box is not checked DeskProto ignores the rotation axis and generates standard XYZ toolpaths. If it is checked, DeskProto will generate XAZ toolpaths instead, so the Y-travel (in linear units) for each tool-position is replaced by an A-value (in degrees). In the View Window the difference will be immediately visible, as the segment no longer is a rectangular block, but a cylinder instead.



When you check “Use Rotation axis”, DeskProto will automatically:
- set the Segment to “Use upper half of geometry”, in order not to let the tool sink below the rotation axis.
- set the Translation to "None" for the Z-axis (resulting in a Workpiece zero point exactly on the rotation axis)
- set the Borders to "No extra" for all operations in that part.
You are free to change these choices later in case needed.
The message box shown above will remind you of these changes, you can switch it off when no longer needed (not yet possible to reset that choice unless by editing the Windows Registry)

Do note that some users prefer a different Translation setting: with the workpiece zero point (Y=0, Z=0) on the top of the cylinder block (Z Translation “Top of block”).

Use 5th-axis
The type of fifth axis that DeskProto expects is a mechanism that tilts the complete rotation axis unit. So in fact this is a B-axis. Note that DeskProto does not control the movement of this axis: it has to be manually set. For more information and a picture se the 5th axis tab of the Operation Parameters.

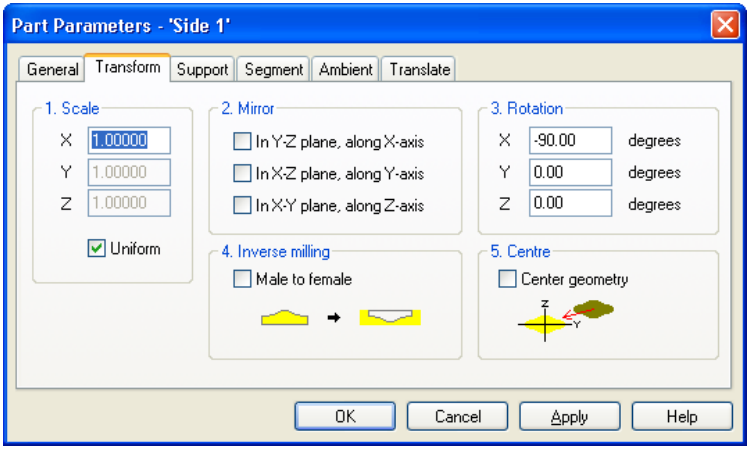
This option is only available in case a machine has been selected that

supports a fifth axis, as defined in the Library of machines This option is not available in DeskProto Lite. If this box is not checked DeskProto ignores the axis and generates standard toolpaths.

Operations
The number of operations within the part can be changed by adding, removing or copying operations, using the buttons **Add**, **Copy** and **Remove**. As three different types of Operation are present (3D, 2D and Bitmap), the Add button comes in three flavours. First select the type of operation using the black arrow button, next press the resulting button to add the Operation you need.
When more than one operations are 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.

Apply
The extra Apply button makes it possible to immediately see the result of any changes, without first having to close the Part parameters dialog. Using Apply also immediately affects the values in the Geometry Information dialog, making it very easy to for instance scale your part until it has certain required dimensions.

Transform parameters



The availability and the contents of options 4 and 5 on this tab page will differ per version of DeskProto (Lite or Full).
Note that the order in which the transformations are applied does matter for

the result, which is why they are numbered in the dialog.

1. Scale
The scale is the first transformation that is applied to the original geometry. It is possible to scale differently for X, Y and Z, by un-checking the **Uniform** option. The imported geometry file does not contain information about the units used. DeskProto assumes the unit used in the geometry file to be the same as set in the Preferences dialog. If this is not true it can be corrected using the scale factor. For instance enter a scale factor of 25.4 to use a geometry file created in inches in a DeskProto that is configured to use metric units.

2. Mirror
The mirror option is the second transformation applied. It is only useful to mirror in one direction, as mirroring in two directions can also be achieved by rotation. Mirroring in three directions can be achieved by mirroring in one direction plus rotation. The mirror option can be useful when you have a geometry that is one half of a prototype. By mirroring the geometry for the second part you can produce two parts that will exactly fit together.

3. Rotation
The rotate option is the third transformation applied. Note the difference between this rotation (which changes the geometry) and the view rotation (which only changes the viewpoint / the camera position). Both rotations use identical values, so you can use the Viewpoint to find the rotation you need, and then use the X, Y, Z rotation values to enter here.
Note that when a custom Segment is present (rectangular) only for rotations over (multiples of) 90 degrees the segment can remain correct. For other angles the segment boundaries will be changed. For Freeform segments only rotations over (multiples of) 180 degrees are possible, other rotations are not permitted

4. Inverse milling
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 in the DeskProto screen is not inverted. To get an idea of what you are machining you can display the rendered Z-grid 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 Segment still is done in un-inverted coordinates.
Inverse milling is not available in DeskProto Lite.

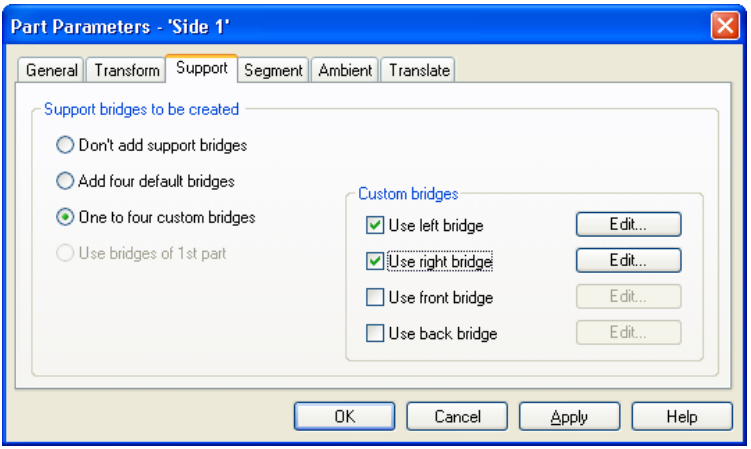
5. Centre geometry
In DeskProto all Translations are applied at the end of the calculation pipeline, after calculating the actual toolpaths. For rotation axis milling this is not sufficient, as the position of the geometry influences the toolpaths that will result.
By default, so when you do not check this option, during rotation axis machining DeskProto will rotate the geometry around the X-axis as defined in the CAD system.
When you do check this option, DeskProto will rotate around a line parallel to the X-axis, on a YZ position in the center of the part. You can immediately see the difference as this influences the size of the cylindrically-shaped segment: fitting the geometry if checked, too large if not (unless of course Y=0 Z=0 is already in the center of the part).
When you need any other position of the rotation axis you have to go back to the CAD system and translate the geometry there.

All three axes X, Y and Z will be centered prior to the calculations. In addition to this option you can still use the Translation options for a translation to be applied just before saving the toolpaths file. 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).

This option is for instance applied when using the N-sided milling wizard.

Bottom level
This parameter is only available in DeskProto Lite, and replaces the (more advanced) segmentation options in the full version.
DeskProto will use a rectangular block as the volume it needs to machine, being exactly the bounding box around the geometry. In DeskProto Lite you cannot influence this block to be machined, however you can choose how deep the cutter may go. For instance for a sphere it does not make sense to machine the lowest half, as the cutter cannot reach that part of the geometry. Instead you will machine two halves and later connect these to become one model. The bottom level can be used to determine which part of the geometry has to be machined. In many cases (like the sphere) you can use the preset option that sets the bottom level exactly halfway down the part. When you select a Custom level you have to enter a Z value in the Edit box.

Support parameters



Support blocks or bridges can be added to the geometry to facilitate machining a model from several sides. These are small blocks of material (cylinder or rectangular), acting as connection bridges to hold the model in place within a frame (that has to be larger than the model). Each support block is created using triangles: these will be added to the geometry, and will be handled by DeskProto just like any other geometry detail.

For more information also see the help pages for the Two Sided Milling Wizard, which show the use of support blocks including some illustrations. Support blocks may be useful in other situations too, for instance a left and a right bridge for fixturing your model on a rotation axis device.

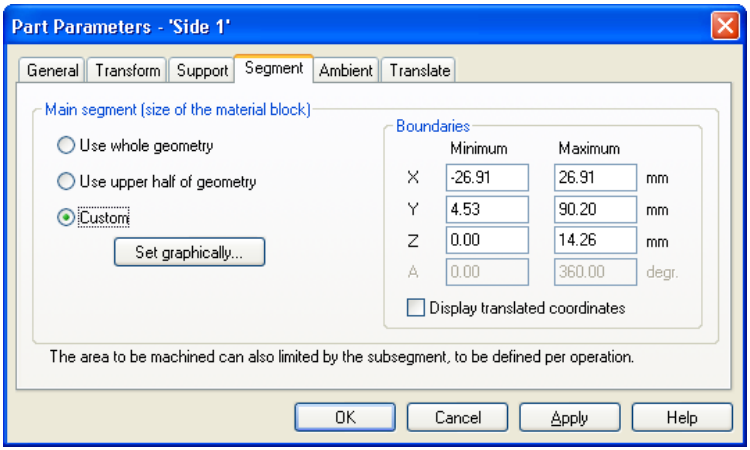
Four options are available:

- **Do not add** support blocks, which is the default and will be OK for most parts.
- Four **Default** support blocks: DeskProto will locate these blocks at the extreme points of the geometry (min and max X, min and max Y, applied after the rotation), and create blocks of a reasonable size.
- When the default blocks do not satisfy, you can choose to create one to four **Custom** blocks. For each of the four bridges you can then check whether or not to use them, and for each checked bridge you can specify location and dimensions graphically, using the **Edit** button. This will open the Edit support bridge dialog, where you can also set the shape of the block.
- **Use bridges of first part** is very convenient when several parts use the same support blocks. Define these support blocks only once (for the first part), and use that same definition in all other parts. In that case also any

changes need to be made only once.

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

Segment parameters



The Part-Segment is in fact the block of material to be used (the stock). It is by definition a rectangular volume, bounded by Min and Max values for each of the three axes. Only the the geometry inside this segment will be processed during the milling calculations, the rest will be removed when machining. In the drawing the bounding box of the segment will be drawn in light brown lines (the green lines that you may see are for the Operation sub-segment). At this tab page you can set the Part segment.
Note that the Min and Max A-values are available only in case you use a Rotation axis.

Use the part segment to machine only a part of your geometry. For instance because a three axis milling machine cannot machine the complete prototype: you can then use segmentation to machine two separate halves. Or because the model is too large for your machine: use segmentation to split it up into parts that do fit.

Three options are given to set the segment boundaries:

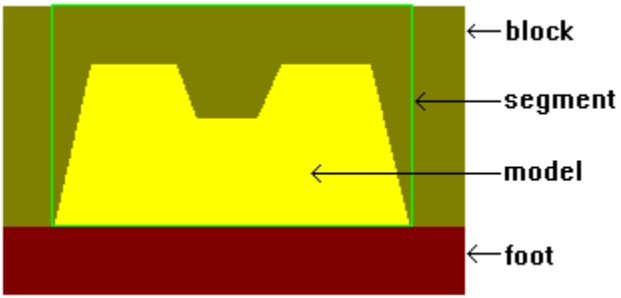
- **Use whole geometry** is the default: the segment will be the bounding box: exactly fit the total geometry. Use this if you want to make a prototype that consists of one part only.

- **Use upper half of geometry** can be very useful when you have a symmetrical geometry, that can be machined in two parts.
- **Custom**. If you want to make your prototype by making parts that are not exactly half the geometry, you can exactly define the segment of the geometry for a specific part. In that case you can either enter the segment dimensions by typing the appropriate minimum and maximum values, or use the button Set Graphically. You can always reset the values by again selecting 'Use whole geometry'. Note that the min and man values for the A-axis are in degrees instead of in mm or inches.

If you want you can enter the values for the segment in **Translated coordinates**. When you have checked this checkbox the minimum and maximum values of the segment (the segment boundaries) position are displayed in the coordinates as used on the machine (after Translation has been applied).
This checkbox also applies to the graphical settings option.
Note:
Checking or unchecking this checkbox does not influence the position on the machine. It's just a temporary conversion on screen for setting the segment more easily.

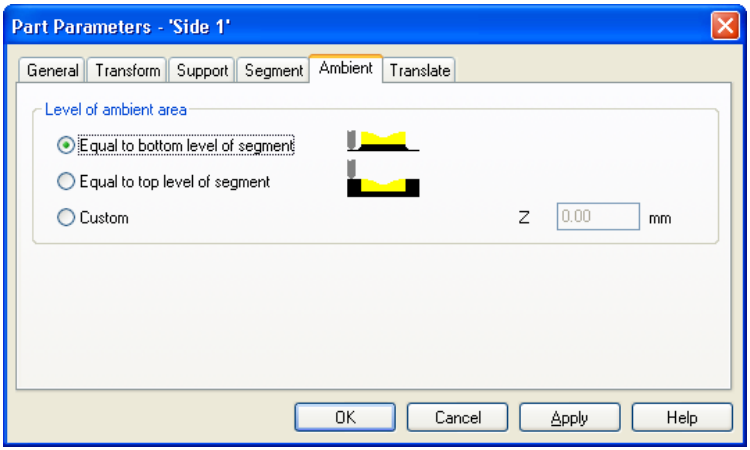
The button **Set graphically** pops up the following dialog that makes it very easy to set any segment using the mouse. This Set segment graphicaly dialog will be explained in the next paragraph.

Note:
You can also use the part segment to make your part larger, adding some more ambient area to be milled around your model. This can be handy for instance if your block of material is too large.
Note:
This larger block trick applies to the Z-direction as well. You can for instance produce a model of 140 mm high, milling three separate 'layer-parts' out of standard tooling board blocks of 50 mm thick, using three segments to define these parts. On the other hand you can also make the segment higher than the geometry, in order to prevent the tool from going too low in a block of material that is too thick for the part. See the illustration below.



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

Ambient parameters



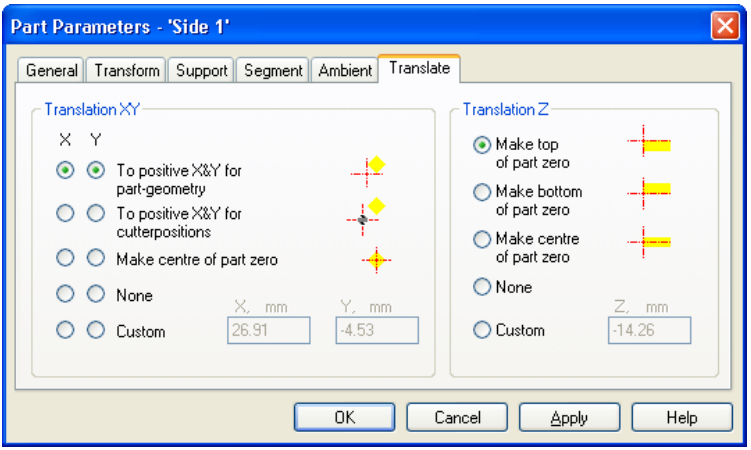
As DeskProto always creates toolpaths for a rectangular area, and as most prototypes do not have a rectangular geometry, some area will be present without geometry. This is called the ambient area. Or in other words, the area where no geometry is present, seen from the top. Here you can specify the level (or height) at which the ambient area should be milled.

- The default choice is **Equal to the bottom of the part**, so at the minimum Z-level of the geometry to be milled. This will do for most cases, as then all material around the model will be removed. When using this option, for ball nose cutters the actual Z-level will be R (radius of the cutter)

lower than the bottom of the segment to make vertical walls possible.

- You can also set the ambient level **Equal to the top of the part**. This will be useful when you want to make a mould by using inverse milling.
- You can also specify the ambient level yourself with the option **Custom**, after which a Z-value has to be entered in the Z edit box.

Translate parameters



As a final part parameter DeskProto will apply an (automatic) translation to your geometry. Although you can see translation as a form of transformation, it's not placed in the transformation tab page. In DeskProto the translation is applied after all actual calculations, just before saving the toolpaths to an NC-program.

Translation is in fact the method that DeskProto offers to set the **Workpiece zero point** for your NC programs. Do note that the Subjects dialog offers an option to draw an Orientator on the workpiece zero point for visual feedback on your Translation settings.

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 of the geometry will be positive (starting at 0), and all Z positions will be negative (starting at 0 as well).

Separate settings are possible for X, for Y and for Z. For each axis three predefined translation options are available (in which DeskProto will

calculate the actual translation values for you, shown in the edit boxes), plus the options None and Custom.

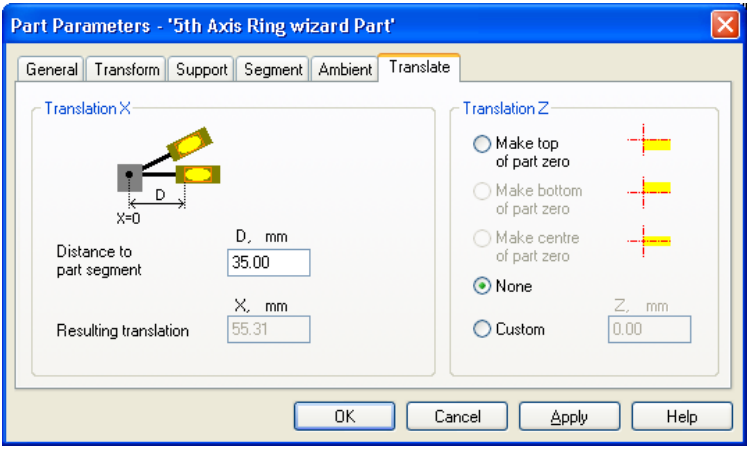
XY:
- You can translate the (transformed) geometry in a way that all geometry points have **positive X & Y values for part-geometry**: the X- and Y-values of the part start at zero and go up. This option is the most convenient one because it will be easy to set the zero-position on the machine.
- You can translate the (transformed) geometry in a way that all cutter positions (or toolpath coordinates) have **positive X & Y values for cutter positions**. 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).
- The third preset option is Make **centre of part zero**: the workpiece zero point will then be exactly in the middle of the part geometry for X and Y.
- **None** means that no translation is applied. The zero-position of the CAD-geometry will correspond with the work-piece zero-point on the machine.
- **Custom** makes it possible to use any own defined translation, to be entered in the two edit boxes. The value that you enter is the translation between CAD coordinates and workpiece coordinates. The edit boxes also show the actual translation for any of the other options.

Z:
- You can translate the (transformed) geometry in a way that all points have negative Z-values. So the **Top of the part is zero**, and Z-values start at zero going down into negative. This option is the most convenient one because it will be easy to set the zero-position on the machine.
- You can translate the (transformed) geometry in a way that all points have positive Z-values. So the **Bottom of the part is zero**, and Z-values start at zero going up into positive. Main advantage of this option is that the workpiece zero point can be the same for parts of different sizes. Do note that in case of ballnose cutter the tip of the cutter can still travel below Z=0 now !
- The third preset option is **Make centre of part zero**: the workpiece zero point will then be exactly in the middle of the part geometry for Z.
- **None** means that no translation is applied. The zero-position of the CAD-geometry will correspond with the workpiece zero point on the machine.
- **Custom** makes it possible to use any own defined translation, to be entered in the edit box. The value that you enter is again the translation between CAD coordinates and workpiece coordinates.

For Rotation axis machining only the X and Z coordinates can be translated here. 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 as they would not make sense.

Note:

When you have checked the option **Use 5th-axis** on the General tab of the Part parameters, a different set of Translation options will be available. See the illustration below.



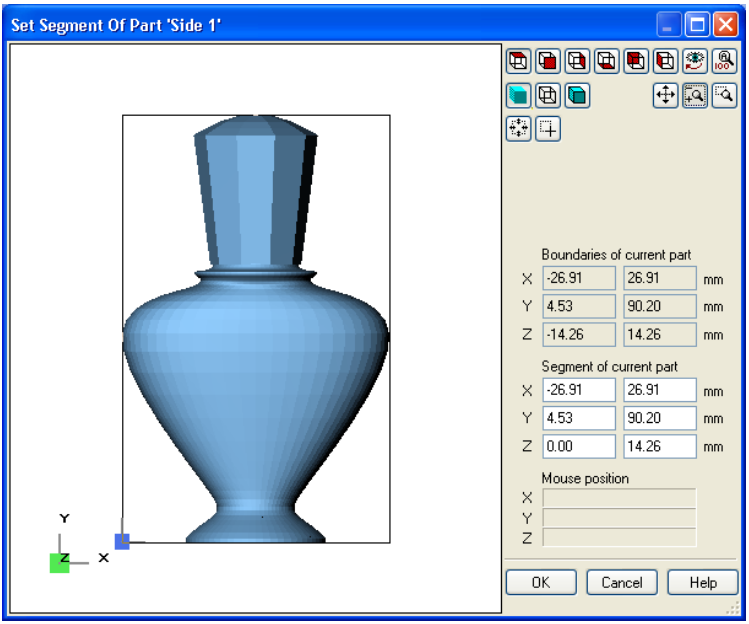
As explained for the 5th axis settings of the Operation parameters, the X-translation will determine the location of the part when the 5th axis rotation has been applied. The workpiece zero point (X=0.0) needs to be exactly on this 5th axis. The small illustration on the dialog shows that a larger distance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

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

A Y-axis translation is not possible here: in most cases 5th axis machining is combined with rotation axis (4th axis) toolpaths, where Y-translation is not applicable.

3.4.5 Set Segment graphically

The Edit segment dialog makes it possible to graphically set the size and position of a (sub)segment. You can reach this dialog by using the button Set Graphically on the Segment tab of the Part parameters or Operation Parameters dialog.



The dialog shows a new drawing of your part, with a rectangle that indicates the (sub)segment that you are editing. The buttons on the right can be used to influence the drawing. Eleven of these are standard DeskProto buttons of which the use is known, and that need no explanation here. Note that mouse rotation is not present: this dialog only used the six main views.
The Cube buttons on the second row are for the choice between **rendered** view, **wireframe** view and **both** views combined.

The two buttons on the third row are new, these are used to graphically set the support block. 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 drag a complete new rectangle to define the segment.

 This button sets the mouse function to resize the current rectangle by dragging each of the four sides to a new position.

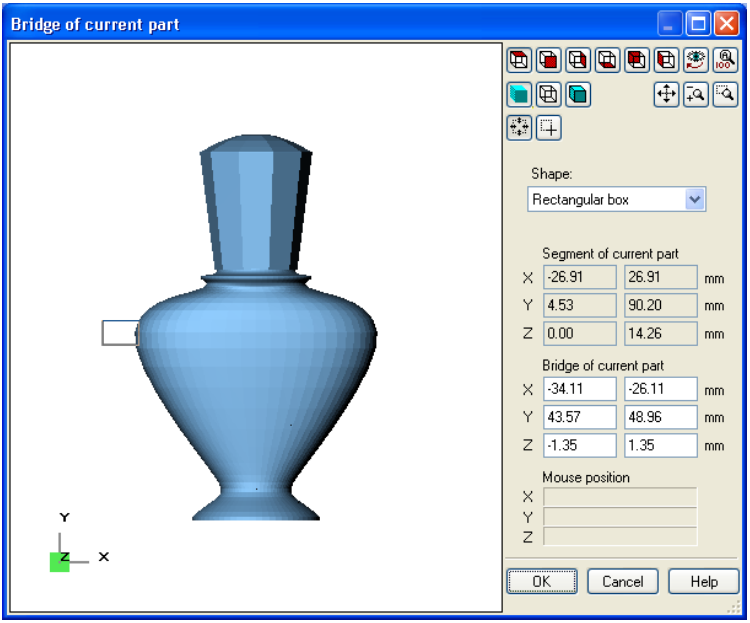
You can also set the new segment by entering minimum and/or maximum values for X, Y and Z in the six edit boxes. The boundaries of the current part are shown to assist you, so are the coordinate values of the current mouse position.

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

As a segment is rectangular and aligned with the main axes, resizing only makes sense in the six main views of your part: 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.

3.4.6 Set support block graphically

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



The dialog shows a new drawing of your part, with a rectangle that indicates the support block that you are editing. The buttons on the right can be used to influence the drawing. Eleven of these are standard DeskProto buttons of which the use is known, and that need no explanation here. Note that mouse rotation is not present: this dialog only used the six main views.
The Cube buttons on the second row are for the choice between **rendered** view, **wireframe** view and **both** views combined.

The two buttons on the third row are new, these are used to graphically set the support block. 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 drag a complete new rectangle to define the support block.


This button sets the mouse function to resize the current rectangle by dragging each of the four sides to a new position.

Blow the buttons you can use the combo-box to select the **Shape** of the support block. Two shapes are available: rectangular and cylindrical, though for cylinder you also need to choose th orientation of the cylinder (only possible parallel to one of the three main axes).

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

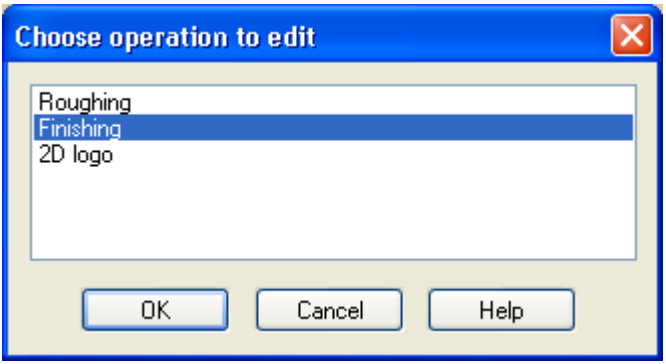
When you change a support block, you will see that both the old and the new block remain visible: the changes will become effective after pressing the OK button.

As a support block is rectangular and aligned with the main axes, resizing only makes sense in the six main views of your part: 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.

3.4.7 Operation parameters command

This Parameters menu command displays the Operation Parameters dialog in which you can edit the parameters of an Operation.
Operations can be either 3D Operations , 2D Operations or Bitmap Operations.

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



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

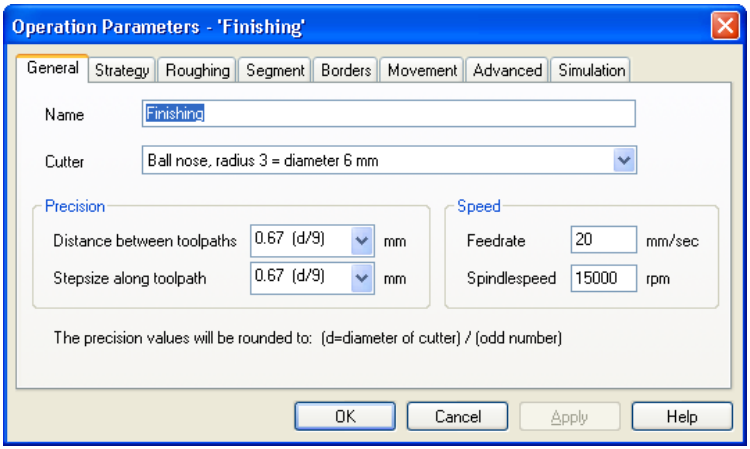
3.4.8 Operation Parameters dialog

The operation parameters are divided into 8 sections by tab pages. The further to the right, the more advanced the parameters in the tab. In DeskProto Lite only the first and the third Tab page are available, as it does offer less parameters than DeskProto Full. Note that DeskProto includes three different types of Operations: this dialog is for the (default) 3D Operation, in addition also dialogs for a 2D Operation and for a Bitmap Operation are available.

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

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

General parameters



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

Cutter
You can select a Cutter from the cutter library using the small arrow button on the right. 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.
The larger the cutter, the smoother the resulting surface and the faster the machining. Use small cutters only in case of small details. You can also use a large cutter for the complete part and a small one later for some detailed areas.

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

to degrees at the outside of the (cylindrical) segment.

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

The precision values used will be rounded to a value that is calculated by dividing the diameter of the cutter by an odd number. The 8 predefined values offered by DeskProto do match this formula. You may enter a custom value as well, which in case it does not match will be rounded to a more precise value that does match. The reason to use this rounding is that the resulting (physical) part will be more accurate, due to the algorithm used by DeskProto for its calculations. For example: When the diameter of the cutter (d) is 4.0, and you enter a precision value of 1.0, it will be rounded 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 are available on the Strategy tab page. These will enable to to use a precision that is higher than the toolpath distance.

Note 1:
In this calculation the diameter of the cutter shaft is used (so not of the tip). When the Collet collision check is applied in the Advanced parameters, the diameter of the collet is used.
Note 2:
Be careful with a Stepsize of D/1 (equal to the cutter Diameter). As the cutter will move in a straight line to the next calculated position, a large step might damage some in-between geometry. This is most likely in case of vertical walls, and can be corrected by the option Protect vertical surfaces.
Note 3:
When a **Skin** is applied in the Roughing parameters, the diameter of the resulting virtual cutter is used. Say you use a ballnose cutter with a 6 mm diameter, and a skin of 0.5 mm. Then DeskProto will do the calculations with a virtual cutter of 7 mm diameter (radius 3.0 + skin 0.5 = 3.5). This means that the first option in the combo-boxes (drop down menu) for Precision now will be "7.0 (d/1)" As this is larger than the diameter of the actual cutter, this first option will be grayed out.
Same for other cutter types, though then the geometry of the virtual cutter will be different (sharp corners will be rounded).

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 part. Do distinguish this speed 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 unit used for the Feedrate are set in the postprocessor of the machine that you selected for your part. DeskProto does not check or even understand these units: it just copies the number that you enter here to the file.

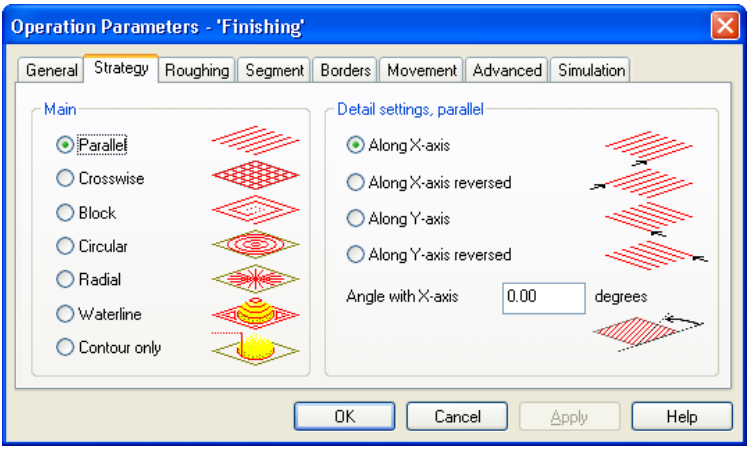
Generally speaking a tougher material will require a lower feedrate. Same for a smaller cutter. A very handy option is to let DeskProto automatically reduce the Feedrate in high Chipload conditions, on Tab page Movement of this dialog.

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

Note:
It is possible to check **Automatic Speed setting** for a cutter in the Cutter definition. In that case when the cutter is chosen the Feedrate and Spindle Speed will be set automatically. That is quite useful for very thin cutters that need a low feedrate and a high spindle speed as otherwise you could forget to set the correct speeds and break your cutter.

Note:
Each Speed option is only available in case your machine supports it, if not the option will be grayed out in this dialog. This setting can be made in the Postprocessor dialog (Options menu).

Strategy parameters



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

Parallel toolpaths is the default strategy: DeskProto will then project a series of parallel toolpaths on the geometry. The toolpath distance is constant (before projection).
Detail settings for parallel are the following: **Along X-axis** with toolpaths on constant Y (so parallel to the X-axis), and **along Y-axis** with toolpaths along the Y-axis. For each of these two starting point are available: front versus back and left versus right. 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.

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. But when you would do that, there would be redundant calculations, resulting in longer calculation time. This option is useful in case the model you want to produce must have a very good surface quality: the staircase effect resulting from the parallel X toolpath will be removed by the parallel Y toolpath 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** can be entered.

The **Block** strategy combines toolpaths parallel to X and Y to a sort of rectangular spiral. These are probably the most efficient toolpaths, for instance very suited for roughing.

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.

Circular is a completely different strategy as here the rectangular base pattern (grid) applied in the first strategies is no longer used. 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 Z-grid. Of course that strategy can very well be used for round geometries, like rings or cups.
Detail settings are:
Inside out versus **outside in** (same as for block strategy)
Yes or no **machine the corners**: the area inside the rectangular segment but outside the largest circle that touches all four sides of the segment.
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 is in the center of your Operation segment, however you can also choose the Center of the Part segment, or any Custom XY values. These custom values then can be either typed or graphically set using the **Set** button. The center point may even be outside the segment.

Radial is the complement of circular: same Z-grid, radial toolpaths perpendicular to circular. So also the same Detail settings do apply, except for the Spiral. And the sequence of the toolpaths now is called **Clockwise** versus **Anti-clockwise**.

Waterline machining produces toolpaths on a constant Z-level (just like the waterlines on a ship's hull). Such strategy is also called contour machining or Z-plane machining. While the difference with toolpaths on constant X or Y seems small, in reality the difference is huge as completely different calculation algorithms are needed.
In the Detail settings an extra parameter is needed: the **Waterline distance** (so the distance between two toolpaths in Z-direction). The XY toolpath distance parameter as set on the General tab page is used when horizontal surfaces have to be machined.
A second waterline parameter is the choice between **Top to bottom**: start at the highest point and work down, and **Bottom to top**: start at lowest Z-level on the outside of the block, and work towards the top.

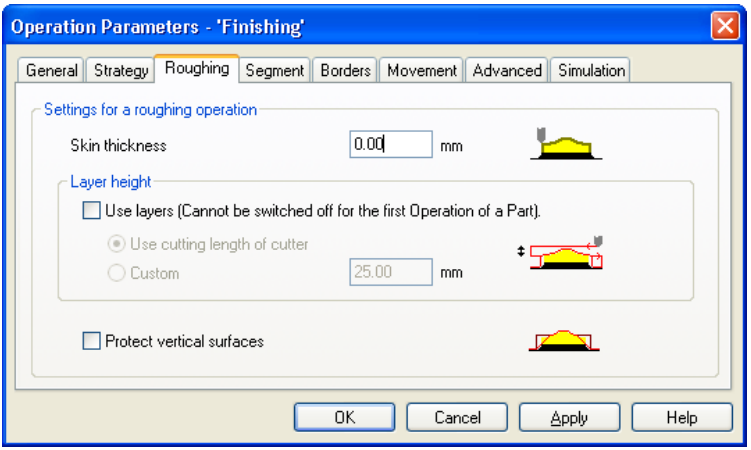
Finally the Detail parameter **Fill horizontal planes is offered**. This option needs some explanation. As waterline toolpaths have a fixed Z-distance in-between each two toolpaths, at (almost) horizontal surfaces there might be a large distance between two toolpaths. This (horizontal) distance might even be larger than the diameter of the cutter: resulting in islands of material remaining after using only that operation. The option Fill horizontal planes checks where the horizontal distance is too large, and fills the space with toolpaths at a distance as specified on the General Tab. All these in-between

toolpaths have the same Z-value, so a visible staircase effect will be the result. You may consider un-checking this option when you use a waterline operation for finishing, after all material has been already removed by previous operations. This will save you much time, both for calculating and for milling.

The last strategy **Contour only** is in fact an additional strategy as it does not machine the complete part: only the outline of the geometry (outer contour) at ambient level is machined. This can be used after a different strategy to smoothen the model (when DeskProto creates toolpaths that are parallel to the X- or Y-axis, the contour of a model at places where it's almost vertical can show a staircase effect). A next use is for Pre-roughing the material to already have the correct outside shape.
No detail settings are available for Contour only.

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 Overrule defaults you can press the Details button. As a result the Calculation Precision dialog will be shown that allows you to fine-tune the gridsize or 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 always need a second operation which machines the same area accurately: the finishing operation. In DeskProto you can use an operation either for roughing or for finishing, so if you need both you will have to **Add** an operation in the General Part parameters.

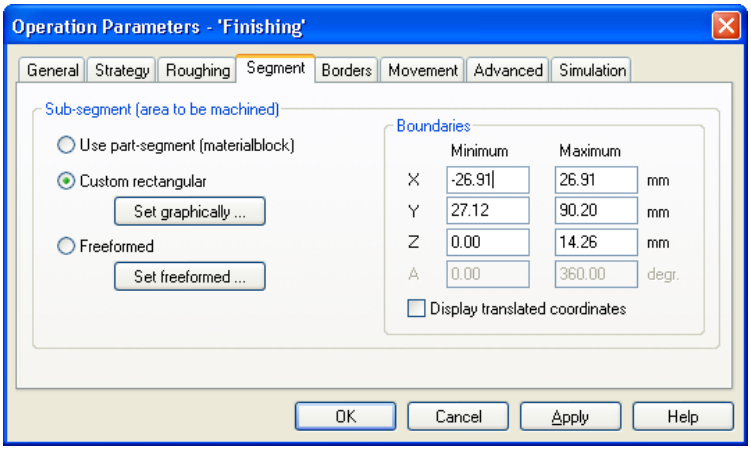
Entering a **Skin Thickness** results in a model which is thicker: a skin is added everywhere around the model. In this way the chance that the cutter takes away too much material is reduced (this can happen as roughing typically will be done using a low precision, and as the cutter will vibrate and may bend during roughing). Using a skin also improves the resulting surface quality, as then during finishing the tool will remove the same 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. This might be 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. Then the cutter machines on full depth, so has to take off the skin of the complete wall in one go. In case the wall is higher than the cutting length of the tool this is a problem for which no solution is available yet. A workaround to solve it is to add an operation using the waterline strategy, milling from top to bottom.

The second roughing option, **Layer Height**, maximizes the cutting depth. The default layer height equals the whole cutting length of the cutter. In most cases it is preferable to use a smaller custom-defined layer height, as with a tough material you do not want the cutter to use its total cutting length. The first 3D operation of a part always uses layers. This is done automatically and cannot be overruled. For subsequent operations you can un-check **Use Layers**: the cutter will then machine at full depth all the time.
Note:
The first layer starts at the top of the segment. When your block is higher than the part you can set a Custom Part Segment with a higher Max Z value.

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. On some machines this may cause unwanted vibrations.
This is in fact the same algorithm as used in Vertical surfaces in the Advanced Operation parameters with a height/step ratio of 1.

Segment parameters



This tab page offers almost the same options as the segment tab page of the Part parameters.
Note that the functionality that is offered is different though ! A Part segment defines the size of the complete Part: any geometry outside will be removed. The Operation segment (also called sub-segment) only defines the **area to be machined**. So it limits the toolpaths for this Operation, for instance when there is a small piece in the model that is very detailed and needs to be machined with a smaller tool in an extra operation. You can use the sub-segment to exactly define an area that you want to machine more precisely in this extra operation. Any geometry outside will not be damaged.

The default option here is **Use segment of part**. This means that the cutter will machine the whole part you want to create. The sub-segment may not be larger than the segment of the part.

Two types of custom sub-segment are available:
Custom Block sets a rectangular block, just as for the Part segment. This block can be defined by entering the min and max boundaries in the edit boxes at Settings, or graphically. The button **Set graphically** pops up a dialog that makes it very easy to set any segment using the mouse, just as is used for the part segment. See the explanation at the Set Segment graphically page of this manual.

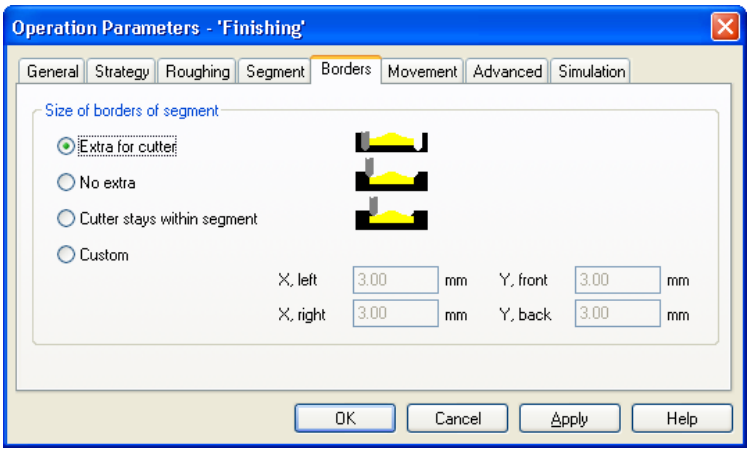
Checking the option **Display translated coordinates** changes the numbers that are shown for the segment boundaries: these will now be in workpiece coordinates as used on the machine, so after Translation. Only the numbers shown here for input are changed, not the actual coordinate values in the

toolpath.
This checkbox also applies to the graphical settings option. Checking or unchecking this checkbox does not influence the position on the machine. It's just a temporary conversion on screen for setting the segment more easily.

The second type of sub-segment is the **Freeformed** segment.This option allows the use of segments that are not rectangular, for instance a circle or any freeform contour (freeformed in top view). For more information see Set freeformed segment.
The Min and Max values shown in the Settings area will be of the bounding box of the segment (for X and Y). The min and max Z can be set as for a block segment.

In case the operation is a bitmap operation a fourth option is present: **Use Bitmap size**. Then the area to be machined will be the bitmap area, so it will not depend on the 3D geometry that is (or is not) present.

Borders parameters



As has been explained already, in DeskProto the toolpaths always form a rectangular area. Normally this area needs to be a bit larger than the minimum and maximum values of the geometry, allowing the tool to move down to minimum Z-level everywhere round the geometry. This extra area at the 4 sides of the rectangle is called the **Border area**, and in this Tab you can influence the size of the border area (the Z-level used for the border area is set at the Ambient tab of the Part Parameters). Note that you can also use the segment to add extra ambient area.

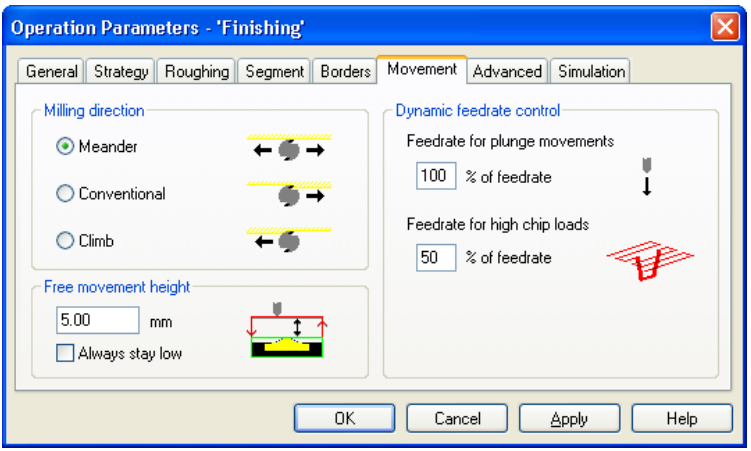
The default option is **Extra for cutter**, which sets the border area size exactly to the size that is needed to let the cutter go around the model, in order to machine all outside surfaces of the part.

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

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

The last option, **Custom**, is 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 should to machine has to remain.

Movement parameters



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

However, on the surface of the model you may see a difference between the

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

The **Free movement height** is the Z-level at which all 'non-cutting' tool movements will be executed. This is for rapid positioning movements over the workpiece when cutting is not being done, 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 geometry. Make sure to enter a positive value: in case of a negative free movement height the model might be damaged.

The free movement level **ZFree** can be used in three ways:

- The first point and the last point of the toolpath for each operation are located ZFree mm/inch above the top of the Part Segment, in order to make sure that the cutter is hight enough when moving to and from these points.
- The positioning moves during an operation are performed at ZFree mm/inch above the top of the Operation segment, in order to speed up the process then machining some detail area at a low level. These movement are done in Rapid mode. When a Skin has been set (Roughing) the free movement level will be the Skin-thickness higher.
- When the option **Always stay low** is checked, the positioning moves are performed at Zfree mm/inch above the highest point of the geometry over which the cutter moves. For machines with a slow Z-axis this will save a lot of time. Also when on yur machine switching between Rapid and Normal movement is slow this will save time, as these movements are done at normal Feedrate.

Note that not all positioning movements are done on Free movement level, as for small distances this is not needed. The following situations are recognized:
1. the distance to be traveled is less than 1.41 times the Precision. In this case the cutter can move in a straight line between the two points (begin and endpoint).
2. the distance is larger than the Precision but but smaller than the Cutter-diameter. Now the cutter will only rise a bit, and travel in a straight line at constant Z-level and at normal Feedrate (speed). The Z-level used depends on the geometry between the two points: ZFree mm or inch above the highest geometry that is found there.

3. The distance is larger than the Cutter-diameter. Now the cutter will rise to Free movement height for a positioning move at Rapid speed (unless of course the option Always stay low has been checked). The idea is that now the distance is so large that a rapid move will gain time.

The **Dynamic Feedrate** control is an advanced option of DeskProto: even many so-called high-end CAM software packages do not offer this type of functionality. It means that DeskProto is able to reduce the feedrate when needed, thus making it possible to select a high overall feedrate without the danger of breaking your tool at the critical points that will be encountered. Two separate options are offered. In both cases you can enter a percentage for the maximum feedrate reduction applied; in both cases DeskProto will choose in-between feedrates whenever possible, thus always running at optimum feedrate. Do note that both options can be combined, in which case for certain movements both reductions will apply, resulting in a very low feedrate.

Using the **Feedrate for plunge-movements** it is possible to decrease the feedrate when the cutter moves downwards. This may be needed to machine 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.
The rate you enter here will be used most downward movements. For downward movements that are almost horizontal 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 by only 60 %.
A line is considered ‘almost horizontal’ when the downward angle (so the angle with a horizontal line) is less than 30 degrees.

It’s also possible to decrease the **Feedrate for high chip loads**, that is when the cutter has to remove much material. Due to DeskProto’s parallel toolpaths approach, normally the cutter only has to remove a small slice of material: a thickness of only the distance between two toolpaths (the stepover). However, in certain cases the cutter has to machine away material over the full flute diameter, which is a much higher chip load. For instance for the first toolpath (as the block will be over-sized in most cases); also when entering a pocket in the model (so when the tool suddenly has to machine much lower than during the previous toolpath). The chip load is even higher in such cases as the chips cannot easily fall out of the cut then, but will be stuck in the groove that is machined.

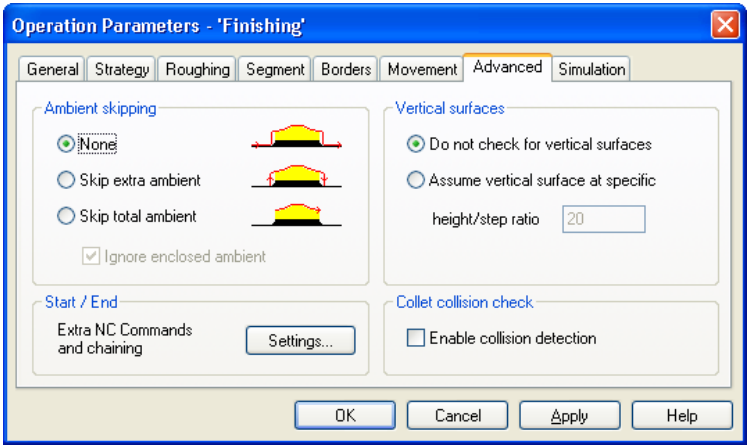
In these cases the feedrate will be reduced, the actual reduction depending on how much deeper the tool has to cut compared to the previous toolpath at that position. Reduction is applied according to the following rule, where D is the cutter’s flute diameter and Rate is the percentage that was entered. 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 D	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 D	Rate	70 %

So 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.

The option Reduced feedrate for high chiploads is not available for all strategies.

Advanced parameters



Ambient skipping concerns the ambient area. Basically DeskProto always machines a complete rectangular area (the segment), due to its parallel

toolpaths approach. In case the model only takes up a small part of the segment, this might lead to many superfluous movements and unneeded 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 **None** (no skipping): the complete segment will be machined.
- Skipping the **Extra ambient** means that all extra horizontal movements on ambient level, from the model to the segment border and back, will be skipped. The tool will still go down to ambient level around the model. You can for instance use this for finishing, when all ambient material has already been removed by the roughing operation.
- Skipping the **Total ambient** means that also the up and down movements (if any) to ambient level around the model will be skipped. The center of the cutter will remain above the geometry: all positions with the center of the cutter above ambient area will be skipped.

An extra options that can be checked here is called **Ignore enclosed ambient** . When checked, only the ambient around the part will be skipped. Enclosed ambient area, like for instance holes in the model, will just be machined.

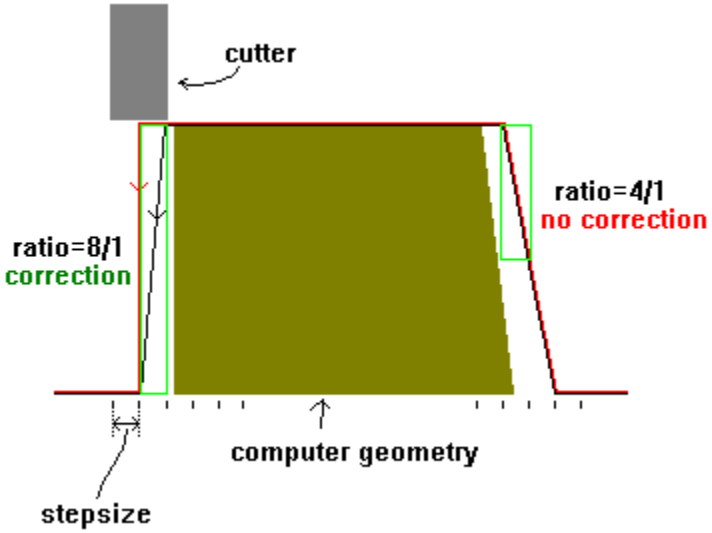
The algorithm used by DeskProto to calculate toolpaths does not really support true **Vertical surfaces**. Each tool-movement will contain both a horizontal and a vertical component, the horizontal one being the stepsize along the toolpath. So true vertical movements are not possible: a vertical surface in the geometry will have a small angle in the model. In case you do need a model with true vertical surfaces you can achieve this 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 example below.

For every movement DeskProto will check this ratio, and if it exceeds the value entered here (so if the movement is steeper) DeskProto will insert an intermediate movement: the movement will be split into a horizontal and a vertical component, to be executed sequentially. The result will be a vertical surface in the model, which otherwise would have been angled. The same algorithm can be chosen in the Roughing tab page, with a fixed ratio of 1 (so for all lines of 45 degrees or steeper).

For freeform surfaces this functionality is not needed, in such case 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. Do take care when creating prototypes with a draft angle: in case this option is used the result might be that all draft surfaces will be made vertical **!**

Example: **Vertical surface ratio = 8**
In the image below you can see 2 toolpaths. One is drawn in black, one in red. The black one is the one that is first calculated. When you use the vertical surface ratio option and set it to 8, the toolpath will be changed to the red one. So on the left the toolpath is corrected whereas on the right it is not, because on the left the height/step ratio is 8/1 whereas on the right the height/step ratio is 4/1. So on the left DeskProto assumes a vertical surface. For a vertical surface DeskProto will add one tool movement, splitting the original angled movement into a vertical and a horizontal part. These two movements will be output in a sequence that leaves extra material on the part.



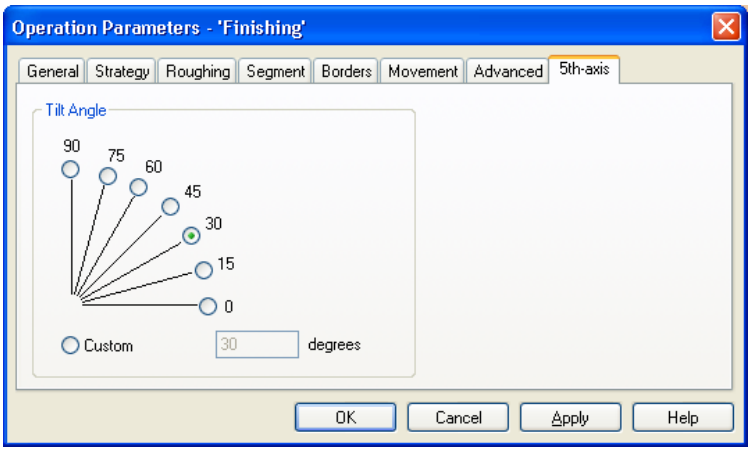
Note that this vertical surfaces check does not work for conical cutters as the steepness of the toolpath is analyzed (for conical cutters the toolpath never can be steeper than the Angle of the cutter). Still checking this option will be effective then: the cutter definition will be changed. The height/step ratio is not used for conical cutters.

Start / End offers the option to add extra activities to the NC program before the operation toolpath starts and/or after is has ended. These activities can be movements, user defined commands or Chaining.
Chaining is an option originally created for the N-sided milling wizard. This wizard combines Operations in separate Parts into one large toolpath file,

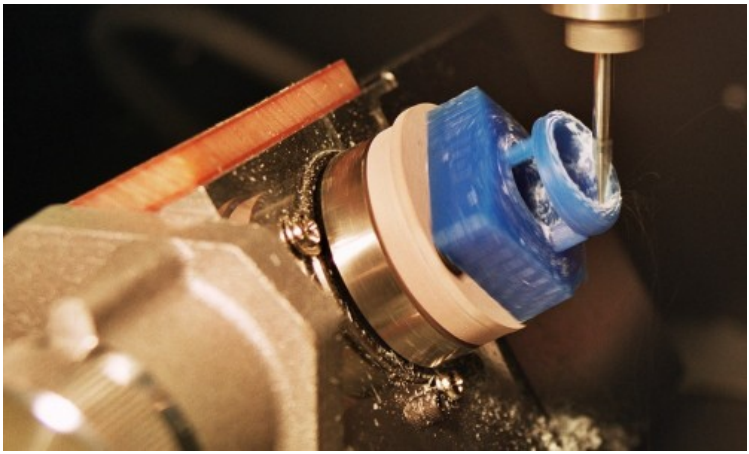
which normally is not possible in DeskProto. Chaining means to connect a next Operation to the end of this current Operation. The Settings button gives access to the Operation Start / End Settings dialog where you can set all of the options just mentioned.

The **Collet collision check** is a very useful option for high models, especially with high vertical or steep surfaces. The problem with such models is that in case the vertical wall is higher than the free length of the tool, the milling machine's collet will damage the model (DeskProto only compensates for the geometry of the cutter). Checking this option means that DeskProto will let the tool move away from the model in such case, preventing the collet to collide with the model. Obviously the resulting model will no longer be correct, as 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 excess material can be removed with a different cutter or by hand.
The diameter of the collet can be defined at the Machine parameters dialog. Do note that when using this option the border area will be enlarged, to enable the complete collet to move down on all sides of the model.

5th-axis parameters



A **5th Axis** is supported by DeskProto only in a special case: the manually controlled B-axis as present on some small wax milling machines for jewelers. Like on the Roland JWX-10 "Jewela" machine, as shown in the illustration below:

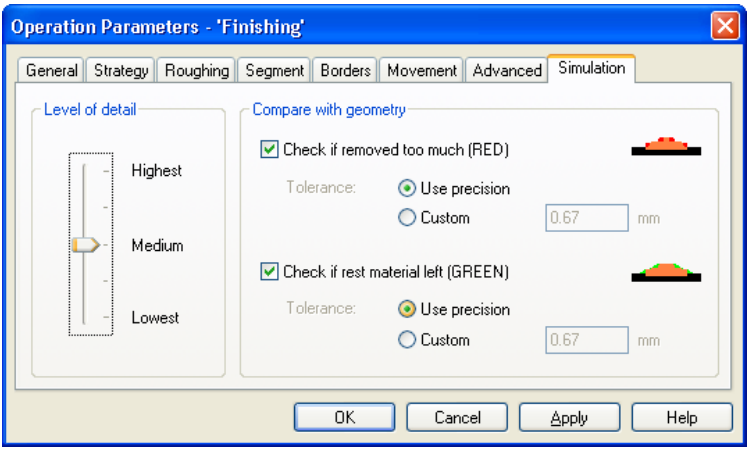


In the bottom-left corner of this picture you can see a rotation knob: after unlocking the rotation mechanism you can use this knob to manually tilt the complete rotation axis (A-axis) unit. This machine supports locking on fixed intervals of 15 degrees.
As this manual rotation is round an axis parallel to Y, this rotation is called a B-axis. The total number of axes then is 5 (X, Y, Z, A and B), hence the name 5th axis.

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 illustation, this rotation offers you the possibility to machine in places where the cutter normally cannot reach: for instance the inside of a ring model.
In addition to these 15 degree intervals you can also enter a **Custom** rotation value here.

Note that the position of the part after this rotation depends on the distance between the part and the actual B-axis (so the axis of rotation). The larger this distance, the more Z-movement while rotating. This distance can be set on the Translation tab of the Part parameters, as at angle 0 this distance is in fact the X-translation. You will see that the Translation tab looks differently when the 5th axis option is selected.
Setting the correct X-translation here is important: only with a correct translation all resulting toolpaths from operations with different tilt angles will be correctly positioned (all with the same Workpiece zero point).

Simulation parameters



A **Simulation** is a drawing on screen that shows you what the resulting part will be. This can be used to check things like the resulting surface smoothness, error movements (if possible) that damage the part, rest-material where the cutter cannot reach, etc. In this dialog you can only set the simulation parameters. You can calculate a simulation by using the Calculate Simulation command in the Create menu. You can also switch the simulation on and off in the Subjects in view Dialog.

DeskProto will calculate a simulation in 3D, so you can rotate, pan and zoom it just like any other subject on screen. Currently the DeskProto simulation can do only one operation at a time, so each operation will have a separate simulation. Also see the description of the command Calculate Simulation in the next chapter of this manual.

In this dialog you can set the simulation parameters, two groups are present:

The **Level of Detail** sets the accuracy of the simulation to be calculated. The simulation is in fact a Z-grid like used in many other DeskProto calculations: the level of accuracy sets the number of grid cells used. The size of one cell is the simulation precision, which is the same as or smaller than the calculation precision.
Five preset options are available to determine how much smaller:

Lowest	Sets this precision the same as the calculation precision
Low	Sets this precision a factor 3 higher
Medium	Sets this precision a factor 5 higher
High	Sets this precision a factor 9 higher
Highest	Sets this precision a factor 13 higher

Note that this Detail factor is applied both on X and Y. So the factor 9 means that per gridcell in the toolpath calculation, 9 x 9 = 81 cells will be used for the simulation calculation. This means that calculating a simulation can take much time, especially for higher levels of detail.

The parameters in **Compare with Geometry** offer 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.

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

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

A **Tolerance** value decides whether or not to apply such color red or green. By default this tolerance equals the calculation precision, though when you set it to Custom you can enter any value that you need in the edit box.

Note: this tab page will not always be visible. DeskProto does not support simulation when the 4th axis is used, so in that case the simulation tab page will not be displayed.

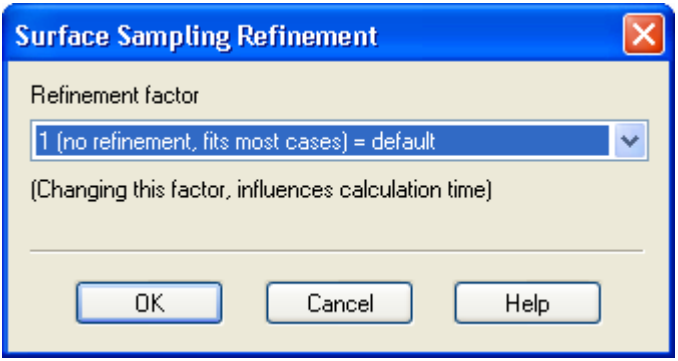
3.4.9 Sampling Refinement

All DeskProto toolpath calculations are based on a Z-grid, and the size of the grid-cells sets the precision of the toolpaths. For every cell in the grid a Z-value will be calculated of the geometry (surface) at that location: the surface is sampled. In other words the DeskProto algorithm is based on **Surface Sampling**.

In the operation parameters you enter this Precision as “Distance between toolpaths” and “Stepsize along toolpath”. The smallest of the two will be used as Gridsize (cellsize) of the Z-grid. While in most cases this works OK, sometimes it is needed to overrule this default and use a smaller Gridsize. This is called subsampling or sampling refinement, as per toolpath point more than one geometry position will be sampled. This dialog can be used to

define the **Refinement factor**.

You can reach this dialog by using the button Details on the Strategy tab of the Operation parameter dialog.
Note that in previous version of DeskProto this dialog was called Calculation Precision Details and had a different layout.



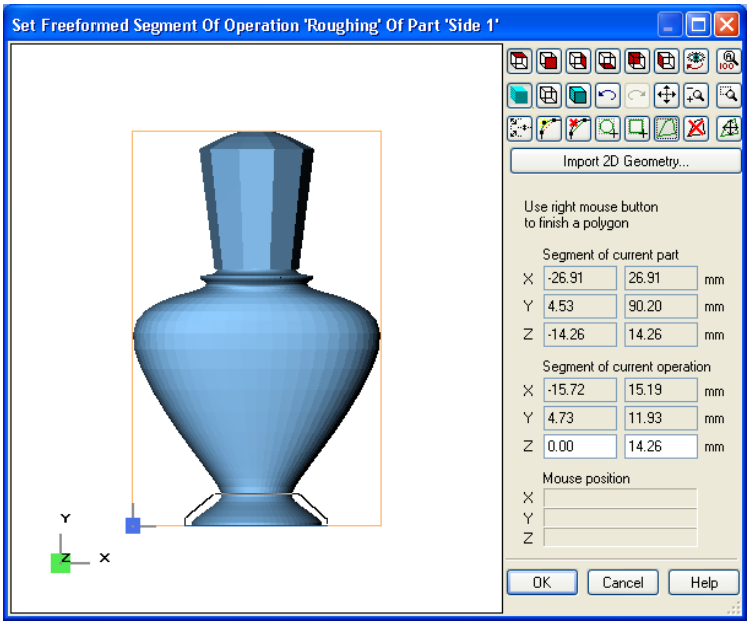
The Surface Sampling Refinement dialog allows you to fine-tune the gridsize or calculation precision. The higher the Refinement Factor, the more accurate the toolpaths and the longer the calculation time (quadratic). A refinement factor 1 will result in a Z-grid with a gridsize that equals the settings for Distance between toolpaths and Stepsize along toolpath. A **Factor 2** will double the Z-grid resolution for both X and Y, so will subdivide each gridcell 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.

3.4.10 Set Freeformed Segment

The Freeformed segment is meant to save machining time by exactly defining the area that needs to be machined. This area can be defined by drawing a freeformed closed contour in the Top view of the part. For this contour you can also set a Min and a Max Z-value, resulting in a freeformed segment. You can reach this dialog by using the button “Set Freeformed” on the Segment tab of the Operation parameters dialog.



The Set Freeformed Segment dialog shows a new drawing of your part, and allows you to draw any closed freeform contour line in the Top view of your Part. The orange rectangle shows the Part segment: the freeformed segment needs to be completely inside this rectangle.
The buttons on the right can be used to create and change the segments contour line. Eleven of these are standard DeskProto buttons of which the use is known, and that need no explanation here. Note that mouse rotation is not present: this dialog only used the six main views. The Cube buttons on the second row are for the choice between rendered view, wireframe view and both views combined.

The eight buttons on the third row are new, these are the drawing tools used to graphically draw and/or change the segment. Note that these are “mouse buttons” as well: of the eleven mouse buttons only one is active at any time.


Edit Freeformed contour: you can move the points and the sides of the contour by dragging with the mouse. Watch the cursor to see if you will drag a point or a line.


Add point to Freeformed contour: click with your mouse to add a point. The closest side of the closest polygon will be split up into two new sides.


Delete point from Freeformed contour: click with your mouse to delete

a point. Watch the cursor when moving the mouse: when a minus sign shows you are on target.


Draw an Ellipse as new Freeformed contour: click the left mouse button, move the mouse and release. The ellipse (can be a circle too) that is drawn is in fact a polyline. When you press the shift button during this input DeskProto will lock horizontal and vertical 1:1, forcing a circle.


Draw a Rectangle as new Freeformed contour: click the left mouse button, move the mouse and release. This function is in fact also present in the Block segment dialog. When you press the shift button during this input DeskProto will lock horizontal and vertical 1:1, forcing a square.


Draw a Polyline as new Freeformed contour: each mouse click will add one point. The polyline is always closed. End the function with a right mouse-click.


Delete Freeformed contour. Two clicks are needed: one to select the contour line to be deleted, and a second to actually delete.


Move Freeformed contour: simply pick and drag the complete contour line.

These buttons are only available in the Top view and Bottom view. In the other views only the Min Z and the Max Z level can be graphically set (these are fixed values, just as for the Block segment). You can also set the new segment'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.

The freeform segment 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.

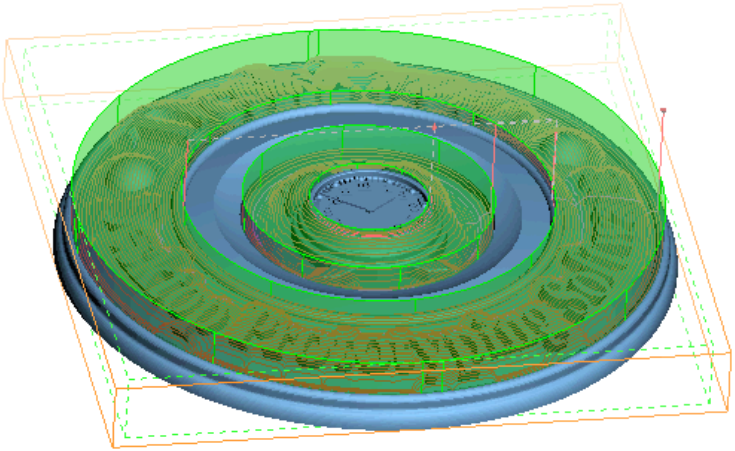

Redo the last action that was undone

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

A completely different way of entering the polyline to define the freeformed segment is by reading it as a 2D DXF file file. The button **Import 2D geometry …** 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 Freeformed segment supports more than one contour: so you can draw more than one freeformed area in this dialog. Nested segments 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.



3.4.11 Set Centre graphically

In order to calculate toolpaths in a circular, spiral or radial pattern a Centre point is needed. Changing the centre point will result in different toolpaths. In the Operation parameters, Strategy tab, Detail settings for these three strategies, you can enter the X and Y coordinates for this centre point. At that same location you can also press the button **Set** in order to open this dialog to graphically set the centre.

The dialog is in fact almost identical to the other dialogs for graphical input: Set Segment and Set Freeformed segment.
Only the functionality is different, and is in fact very limited: click a point to set the XY coordinates for the centre point. This point may also be located outside the segment.

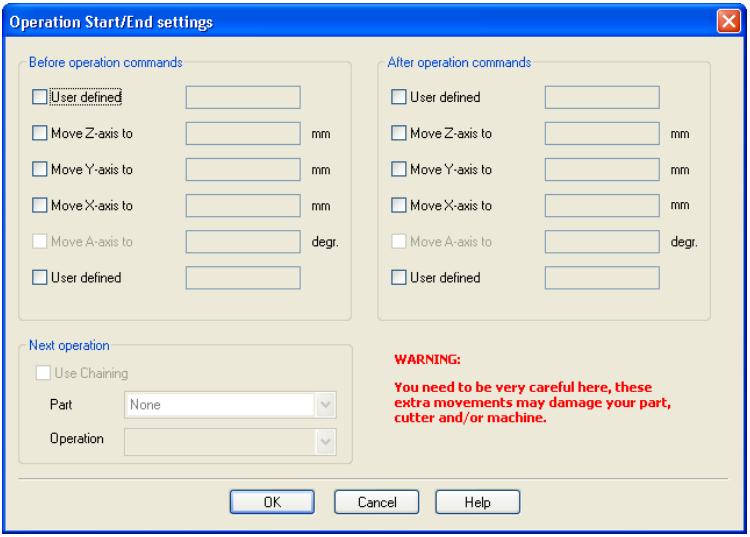
Note that this only works correctly in Top View and in Bottom View: in any of the other views only one of both coordinates is changed when clicking.

3.4.12 Operation Start/End commands

This dialog is meant for advanced users only, as you can conclude from the warning in red text.
It enables to to add extra command lines to your NC program file, both at the start of the file and at the end. For instance to let the cutter travel to some safe position after finishing the operation. Or to let the A-axis rotate to a certain angle before starting to machine the operation. And in addition you

can also specify chaining (see below) in this dialog.

You can reach this dialog by pressing the button "Settings" for start/end on the Advanced tab of the Operation parameters dialog.



Up to six **Before Operation commands** (or Start commands) can be specified. These will be written as extra lines in the NC program file, just before the toolpath of this operation starts. Each line is optional, and will only be written when checked (checking none will mean that no in-between lines are inserted) :

- **User defined** can be used to issue any other command, like moving some other positioning device. Note that the line will be written exactly as defined in this edit box: so take care what you enter ! So you need to know the language that your machine needs: this line will not be translated by the postprocessor.
  We advise that this option be used only by advanced users !!
- **Move Z-axis to** adds a Z-movement command to the NC file: to the specified position (in mm or inches) in workpiece coordinates. Note that these lines are written to the file in the same sequence as present in the dialog: first the Z-movement, and then the X, Y and A movement.

Note: the four optional Move commands are not available for some machines. A machine that always needs three (or four)coordinates per command without specifying which is for X, Y or Z cannot accept a line with for instance only a Z-axis coordinate as it won't know for which axis it is meant. For such machines all Move commands will be grayed out.

- **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 rotation axis to Angle degrees. This option is being used by the N-sided milling wizard: milling from four sides will involve four operations, each time with a 90 degree rotation in-between.

Note: this optional command is available only when the machine for this part has a rotation axis, if not it will be grayed out.
- **User defined 2**, see the explanation given above.

The six **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.

Chaining is an option meant to combine Operations in different Parts into one large NC Program file. Operations in the same Part will be combined into one NC Program file anyway. This option is used by the N-side milling wizard.
Note that chaining is of course only available in case more than one Operation is present in the current project.

The Operation Chaining Settings dialog gives you the option to chain a **Next operation** to the current one. As a result, when saving the toolpath file (NC program file) for the current operation, when finishing it will continue with the next chained operation , writing the toolpath to the same file. In order to select a next operation you have to select both a **Part** and an **Operation** (from that part). Note that operations that already belong to a chain cannot be selected.

Note that if the Operation is part of a Chain a new button will appear in the Operation Parameters dialog: Apply to Chain. Using this button you can apply a new parameter setting (for instance a different precision) to all operations that are in the chain. The alternative is button Apply (or OK) that will only change the current Operation.

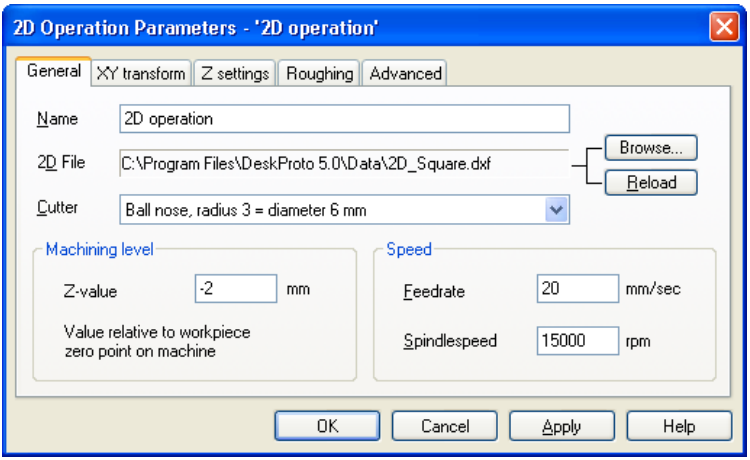
3.4.13 2D Operation Parameters dialog

The 2D Operation parameters are divided into 5 sections by tab pages. Note that DeskProto includes three different types of Operations: this dialog is for the 2D Operation, in addition also dialogs for a 3D Operation and for a Bitmap Operation are available.

This same dialog is used for the Default 2D Operation parameters, only with an extra button **Restore DeskProto defaults** to reset the original default

parameters.

General parameters



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

2D File
With a 3D Operation the toolpaths are calculated using the 3D geometry that has been loaded. With a 2D Operation the situation is different: DeskProto lets the cutter move along the lines of a 2D drawing. The option 2D File allows you to select which drawing file (or plotfile) to use. So every 2D Operation has it's own set of 2D contours.
Use the **Browse** button to pop up a File-open window that you can use to select the 2D file that you need. The **Reload** button can be used to later reload this file if needed.

For 2D files DeskProto supports a DXF format subset and an EPS format subset, both of which includes points, lines, polylines and arcs. All in 2D: any Z coordinate in the file will be ignored. Currently DeskProto will convert arcs to polylines. A point in the 2D file will result in drilling a hole on that location.

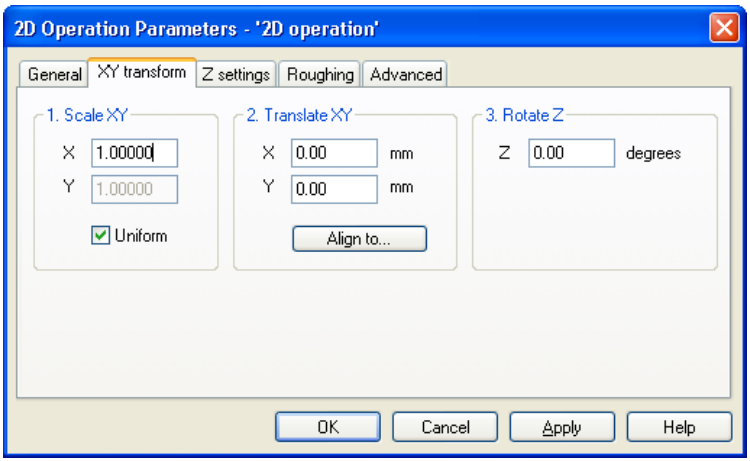
Cutter
The tool that you want to use is one of the parameters that may be different

for different operations. You can select a cutter from the cutting tool library using the small arrow button at the right. Adding new cutters to the library, changing an existing cutter or just retrieving information on a cutter can be done using Library of cutters (Options menu).
Note: for 2D machining DeskProto currently does NOT compensate for the radius of the cutter. The cutter will just follow the contours in the 2D file.

The **Machining level** is the Z-coordinate for the actual toolpath. You can describe this as the *pen-down level* when plotting the 2D file. The Z-coordinate that you enter has to be in workpiece coordinates, so in the coordinate system used on the milling machine.
Note that when the 2D contour is projected on the 3D geometry (see below) then this machining level is taken relatively to the geometry at that point. So a value of –0.5 mm will create a groove of that depth over the 3D surface.

The **Speed** settings are identical to the settings offered on the 3D Operation Parameters dialog: see the explanation on **Feedrate** and **Spindle Speed** there.

XY Transform parameters



The Transform parameters for 2D are meant to correctly position the 2D plot data over the 3D geometry.
2D files are imported in workpiece coordinates, so in the coordinate system as used on the machine. Importing in CAD coordinates would not make sense, as for instance rotation around the X-axis is not allowed for 2D data.

This implies that 3D CAD data and 2D CAD data from the same CAD

system are not automatically aligned in DeskProto !! For a correct alignment:
- do not use any scaling or other transformation for 3D nor 2D, and
- in the Part Parameters select “None” for the X and Y Translation settings.

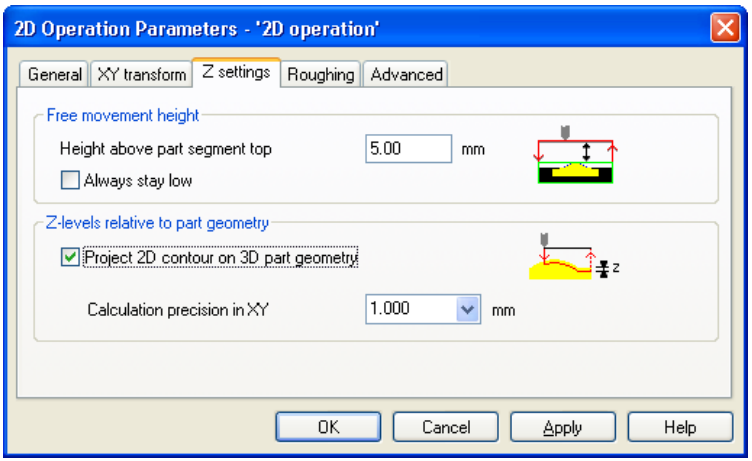
Scale XY can be used to scale the 2D data (the plotfile). It is possible to scale in only one direction, by un-checking the uniform option.

Translate XY can be used to move the 2D contours along the X and/or the Y-axis. Translation values have to be given in mm or inches.

Rotate XY can be used to rotate the 2D contours. Rotation is only possible in the XY plane, and will be done round the zero point of the 2D Drawing.

For correctly positioning the 2D contours over the 3D geometry the **Apply** button proves to be very handy as you then can see what you have done so far.

Z Settings parameters



The **Free movement height** is the same as in the 3D Operation: it specifies the height at which the cutter can freely move without touching material. This height is used to move the cutter from one position to another without machining (pen-up level). It is set as the height above the top of the segment of the part (as a 2D operation does not have a subsegment). Negative values are permitted here. 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*

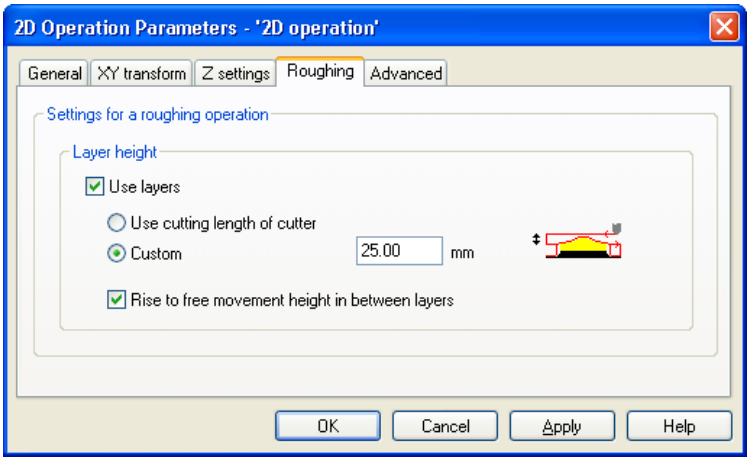
level.

The option **Always stay low** also is the same as in a 3D Operation: positioning moves now will now be done in rapid mode on free movement height, but instead as normal movement, as low as possible in order not to damage any geometry.

The option **Z-levels relative to part geometry** (not available in DeskProto Lite) does in fact project the 2D contour on the 3D part geometry. When you check this option, the Machining level as specified on Tab 1 (pen-down level) is no longer interpreted as a standard Z-level, however is taken relatively to the Z-level of the geometry at that point. So a level of –1 mm results in a groove of that depth over the 3D part. This option is ideal for instance to engrave a logo or text onto a 3D design: the 2D plotdata will be converted into a 3D toolpath.
This option is only available if there is a 3D geometry loaded, and at least one 3D Operation is present before this 2D operation. This 3D Operation makes sure that the material above the geometry has been removed.

In order to calculate such projected 2D toolpath DeskProto needs to make a Z-grid. The option **Calculation Precision in XY** allows you to set the precision, so the grid-size) of this Z-grid. The smaller the value entered, the more accurate the toolpath will be, and the more calculation time needed. This Z-grid will be calculated only for the area covered by the 2D file.

Roughing parameters



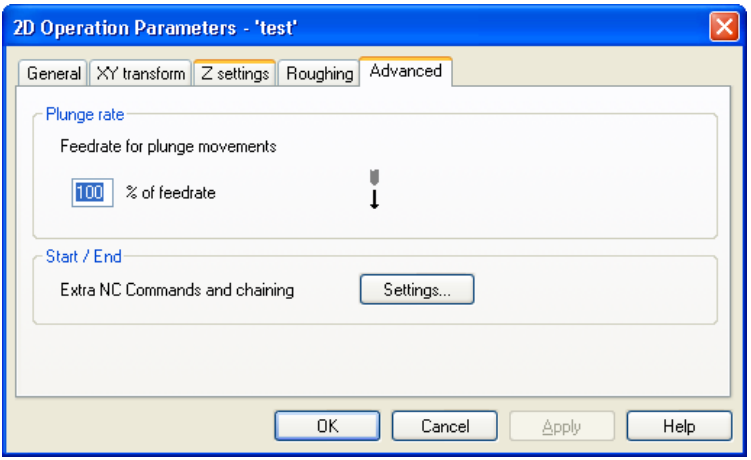
The Roughing settings basically are the same as in the 3D Operation: going

down in layers as the cutter is not allowed to go to full depth in one go. When you un-check **Use Layers**, then the cutter will immediately go down to full depth.

The **Layer height** that is specified determines how deep the cutter may go into the full material. You can set this Layer height to the Cutting length of the cutter or to a Custom value. The custom value may not be higher than this cutting length though.
Rise to free movement height in-between layers does exactly what it says. After completing a layer the cutter will go up, travel to the start point of the contour if needed, and then go down to the next layer. All toolpaths will be done in the direction as set in the 2D drawing software that created the DXF file.
Un-checking this option saves machining time as the cutter will lower to the next layer without going up first. For open contours this will result in meandering toolpaths: go from start to end, then go down to the next layer, travel from 'end' to 'start' etc.

As said before a point results in a drilling operation. Using layers for such operation results in **Peck drilling**: at regular intervals the cutter will move up to get rid of chips.

Advanced parameters



Most of the Movement parameters and Advanced parameters of the 3D Operation do not apply for 2D, only these two settings are useful. The meaning of each setting is the same as in a 3D Operation, so please read that help information:

For **Plunge rate** (moving slower for downward movements) see the 3D Movement parameters.

For **Start / End** (extra command and chaining) see the 3D Advanced parameters.

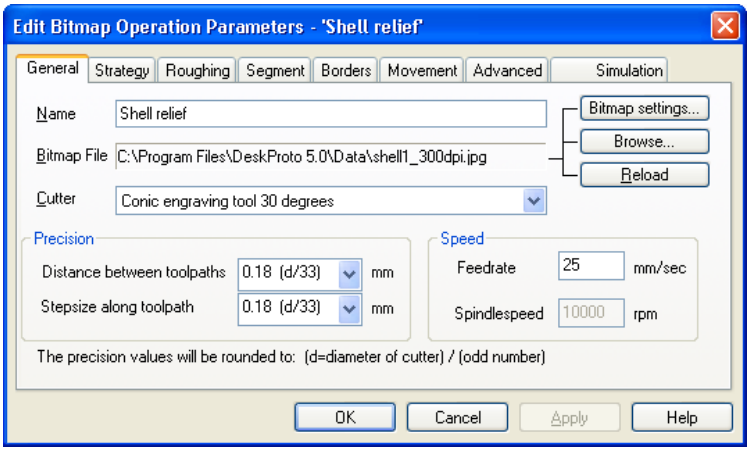
3.4.14 Bitmap Operation Parameters dialog

The Bitmap Operation parameters are almost the same as the 3D Operation parameters, as for both operations the toolpaths are made for a 3D geometry. The Bitmap Operation uses the Bitmap Geometry, the 3D Operation uses the Part geometry. The only differences are some extra options on tab page General, on one extra option on the segment tab.

This Help page only explains the extra options for the General Tab page, for all other options the Help can be found in the 3D Operation parameters help.

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.

General parameters



Name, Cutter Precision and **Speed**
For more information see the the 3D Operation parameters.

Bitmap File

With a 3D Operation the toolpaths are calculated using the 3D geometry that has been loaded. With a Bitmap Operation the situation is different: DeskProto uses a bitmap file to calculate a Relief: the Bitmap geometry, and uses that relief to calculate the toolpaths.
Use the **Browse** button to pop up a File-open window that you can use to select the Bitmap file (bmp, jpg or gif) that you need. The **Reload** button can be used to later reload this file if needed.

The button **Bitmap settings...** will open the Bitmap Settings dialog, in which you can set parameters for converting the 2D bitmap image to a 3D Relief.

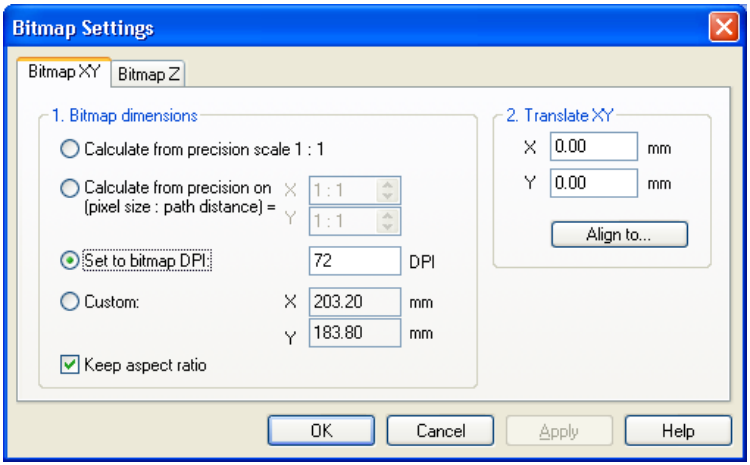
Note: the Precision chosen in this dealog may influence the size of the resulting relief.

3.4.15 Bitmap Settings dialog

In order to use a Bitmap image in DeskProto it needs to be transformed to a 3D Relief: the Bitmap Geometry. In this Bitmap Settings dialog you can set the parameters for this transformation. The dialog can be reached via button Bitmap Settings... in the Bitmap Operation parameters dialog. You can also open it by doubleclicking the line of the Bitmap Operation in the Tree while keeping the Control button pressed.

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

Bitmap XY



Bitmap dimensions
The dimensions are in fact the size of the Relief in XY. As explained in paragraph Bitmap Geometry, DeskProto uses a Z-Grid to create the Relief. In standard 3D Operations the size of one Z-Grid grid-cell is determined by the Precision values in the Operation parameters. In choices one and two this grid-cell size is used to set the dimension of the relief.
Note: For options 1 and 2: when you change the Precision, the size of the Relief will change as well !

- **Calculate from precision scale 1:1**. One pixel in the image will be one grid-cell in the Z-grid, with dimensions as set in the Precision for this Operation.
- **Calculate from precision on scale** =. 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 / Cell size. The higher this Ratio, the larger the relief dimensions. For instance a Ratio 2:1 means two Grid Cells for every pixel, doubling the dimensions.
- **Set to bitmap DPI**. Some bitmap files contain information about their size: the DPI value (bmp and jpg files may contain DPI info, gif files do not). DPI sets how many Dots Per Inch must be taken. So for a 300 DPI image, the size of one pixel will be set to 1/300 inch. The number of pixels used for one grid cell will be determined by the Precision.

When the bitmap file does not contain DPI information this option will be grayed out

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

Note: For options 3 and 4: the ratio between Pixel size and Grid cell size is

not necessarily a whole number. So it might be that some Grid cells 'contain' more pixels than others. This may lead to a **Moire pattern** like a small ridge every few mm. When this happens choose option 2 and approximate the requested dimension as close as possible.

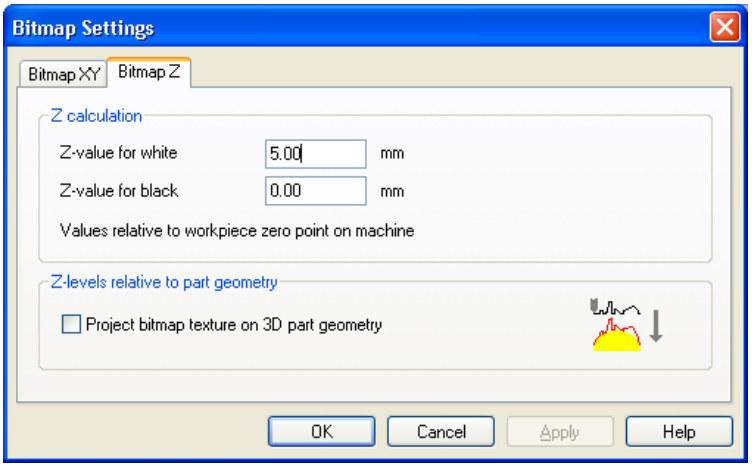
The option **Keep aspect ratio** takes care that the ratio between X size and Y size is not changed: same ratio for both image and relief. So when you change the X size with this option checked, you will see that the Y size automatically changes too.

Translate XY
The position of the bitmap relief relative to the workpiece zero point can be changed here. By default the relief is located with it's lower left corner exactly at XY (0,0), so at the zero point (the Z depends on the settings of the second tab page). If this is the correct location you can change it by entering Translation values for X and Y here.

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

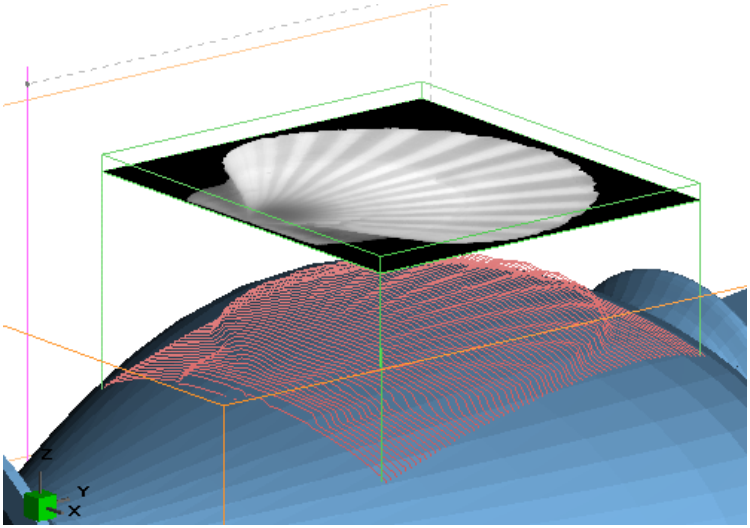
This translation is valid only for this Bitmap Operation. Note that the Translation and the Transformations is the Part parameters do not influence the Bitmap relief. So if you want to align the Bitmap relief with the Part geometry it might be easy to select "None" for the X and Y Translation settings in the Part parameters.

Bitmap Z



Z calculation
As explained in paragraph Bitmap Geometry, the 3D relief is created by calculating a Z-height for every pixel in the bitmap image. Minimum Z for black pixels, maximum Z for white, and the lighter the higher for gray values (or of course the other way round). In these two edit boxes you can enter the Min and Max Z-values, to be used for black and white.

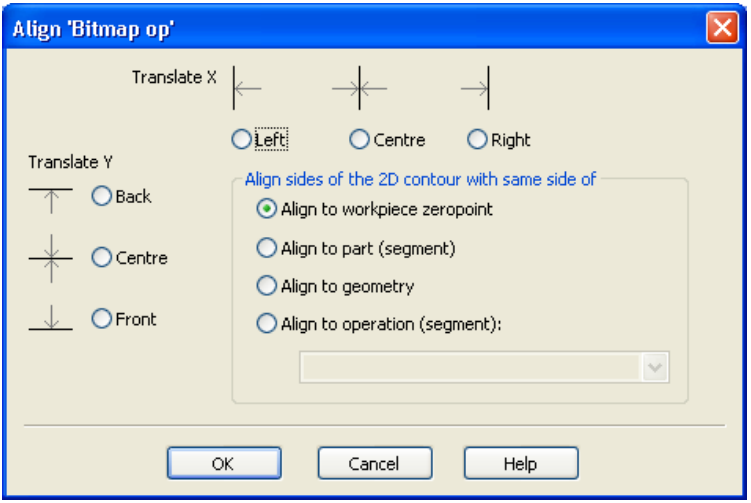
Z-levels relative to part geometry
When the option **Project bitmap texture on 3D part geometry** is checked, the Z-value that is calculated for the color of that pixel is added to the Z-value of the Part at that location. So the 3D relief is modeled around the underlying geometry, using a vertical projection. See the illustration below.



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.
Note 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, using the same bitmap file and bitmap settings (use Copy Operation).

The relief to be projected may also be negative: when the Z-values range from 0.0 to a negative value, the relief will be subtracted from the part geometry.

3.4.16 Align Operation



This dialog is used for 2D Operations and Bitmap Operations, and can be reached using the "Align to..." button in the 2D Operation parameters and the Bitmap Settings. As these types of Operations both use their own geometry file, it may be needed to align this geometry with the 3D Geometry as defined by the Part. This dialog makes such alignment easier.
Only alignment by Translation for X and Y is supported, Z Translation and rotation are not possible here.

The thing to be aligned is the **Bounding box** of the 2D contours (for 2D Operations) respectively the **Bitmap image** (for Bitmap Operations). Also in case the Operation segment has been made larger.

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

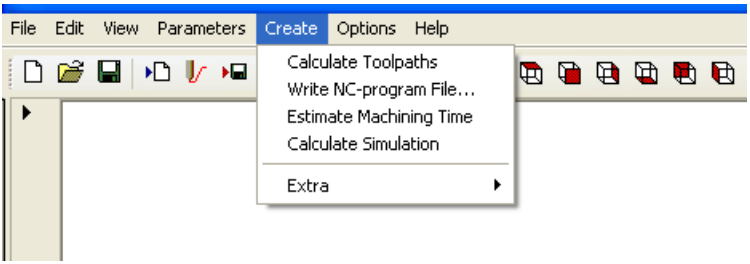
- **Left** means that the Left side of the Bounding box / Bitmap image will be aligned with the entity (for the WP zero point) or with the left side of the entity (for the other three).
- **Center** means that the Center side of the Bounding box / Bitmap image will be aligned with the entity (for the WP zero point) or with the center of the entity (for the other three).
- **Right** means that the Right side of the Bounding box / Bitmap image will be aligned with the entity (for the WP zero point) or with the right side of the entity (for the other three).
- **Other** means that the current translation value fits none of the above. It will not be changed then.

For **Y-Translation** the options are in fact the same as for X.

The Bounding box / Bitmap image will be aligned with one of four entities:

- **Align to workpiece zero point**. Always possible.
- **Align to part**. The part segment is used. Only available in case a geometry has been loaded.
- **Align to geometry**. The bounding box of the geometry is used. Only available in case a geometry has been loaded.
- **Align to operation**. The operation segment is used. Only available if the Part contains other operations. In case more than one you need to choose which of these operations to use.

3.5 Create Menu

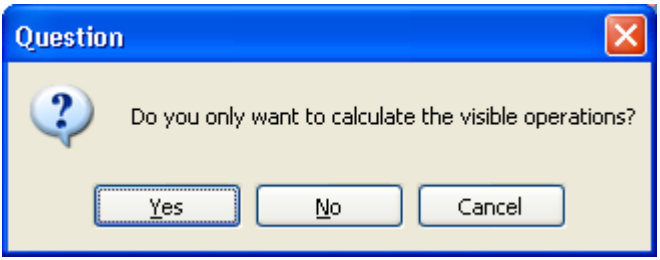


These options control all actions for calculating and saving NC toolpaths. Note the submenu called Extra which offers some extra options. As these are very specific and not important for most users these commands have been "hidden" in a submenu.

3.5.1 Calculate Toolpaths

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

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



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

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

Shortcut:

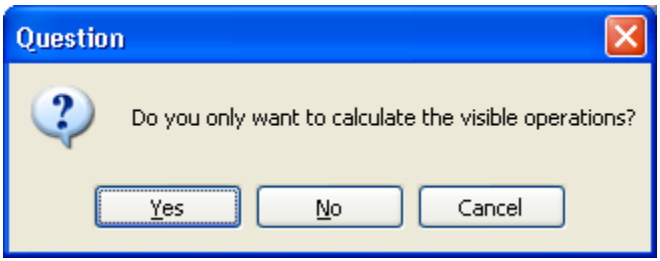
Toolbar: 

3.5.2 Write NC program

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

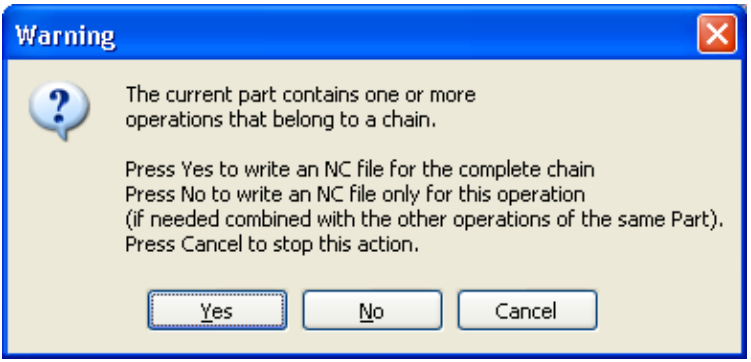
First you will be asked to give a name for the NC-program file, in a standard Windows Save-As dialog, in which the correct file-extension for your machine has been already entered. The only thing you have to do are make sure the file is being saved in the right place and choose a filename.

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



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

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



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

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

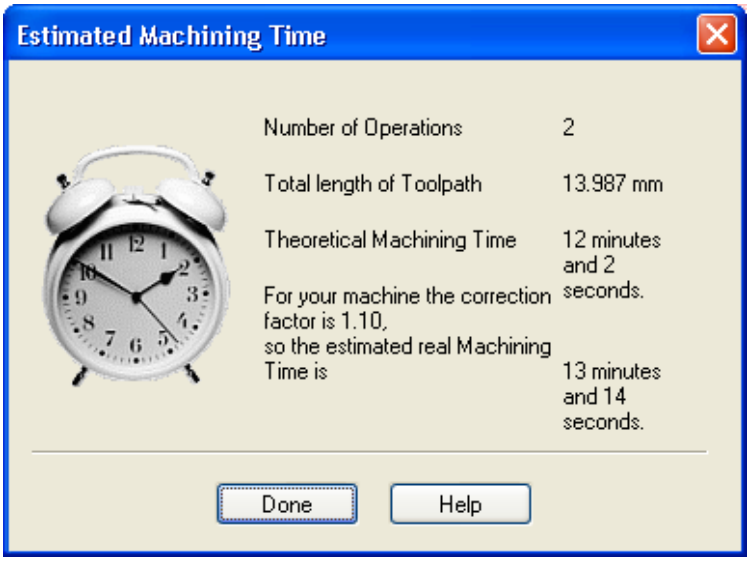
The name for the NC file that you enter here might be automatically changed. When there are more than one Operations in the part or a Chain is involved, and when different cutters are used in the Operations, and when your postprocessor start a new NC file at a toolchange, then two or more NC files are written. Names for these subsequent files will be automatically generated. In case you chose the name Test.nc and the two operations were called Roughing and Finishing, then file first file will be called Test#1_Roughing.nc and the second file Test#2_Finishing.nc

Shortcut:

Toolbar: 

3.5.3 Estimate machining time

With this option you let DeskProto calculate a rough estimation for the machining time that it will take to create your part. Do note that this is a very rough estimation: the actual time will be influenced by many factors.



Theoretically speaking the calculation is very easy: DeskProto knows both the length of the toolpath and the feedrate (machining speed), so length divided by feedrate results in a theoretical machining time. However the real machining time is influenced by the answers to questions like:

- How long does the machine's controller take for linear interpolation calculations (calculating the separate speeds for each axis, needed for every movement) ?
- Does the machine keep up its speed, or does it stop in between each two movements ?
- How fast can the machine accelerate and decelerate ? (especially important when stopping or slowing down after every movement).
- How fast is the data transfer from computer to machine ? (if you are using a 9600 baud serial line this factor will seriously slow down the process).
- Does the toolpath consist of small movements or long straight lines ? (in the first case it will in fact not even reach the desired feedrate as the distance is too short to accelerate to full speed).

So the resulting real machining time **cannot even be correctly predicted for one particular machine**, as it will vary considerably depending on the characteristics of the geometry.

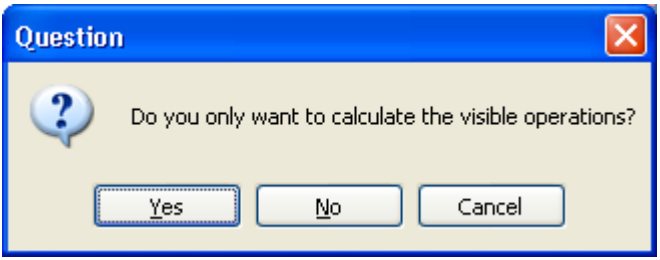
DeskProto will convert the Theoretical machining time to an estimated Real Machining Time by multiplying by a machine dependent correction factor. As previously noted, this results in a rough estimation since the actual time is also dependent on the toolpath characteristics. The correction factor can be

set at the Machine dialog (Library of Machines). In order to fine-tune you can time a few toolpaths and correct this factor accordingly.

3.5.4 Calculate Simulation

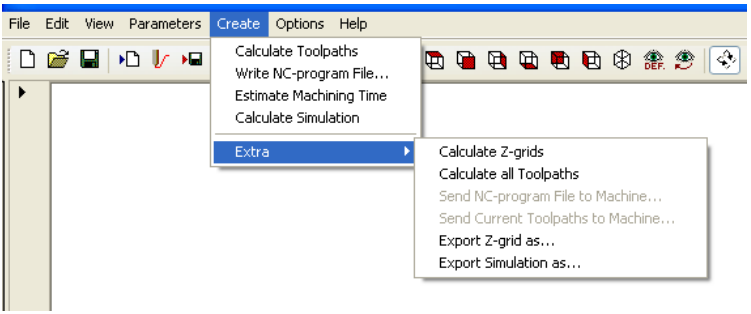
With this option you can create the Simulation for the current part.

In case all operations of the current part are visible (see Visible Operations) in the active view, all the Simulations 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 the visible operations for calculations or if you want to use all operations.



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

3.5.5 Extra submenu commands

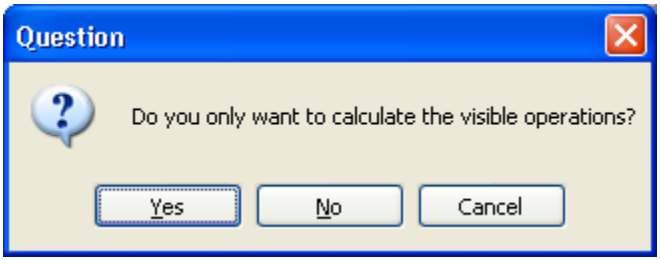


These Extra commands also are options for calculating and saving NC toolpaths. As these Extra commands are very specific and not important for most users these commands have been "hidden" in a submenu.

3.5.6 Calculate Z-grids

With this option you can create the Z-grids 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 the visible operations for calculations or if you want to use all operations.



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

3.5.7 Calculate All Toolpaths

With this option you can create the 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. This command has been added mainly for internal use, with chained operations as used in the N-sided milling wizard

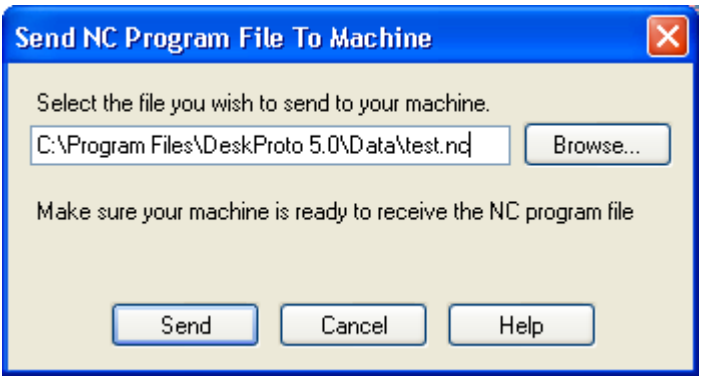
Do note that only the visible (see Visible Operations) operations of each part will be calculated. In case one or more invisible operations are present these will be ignored, so you are NOT asked if you only want to use the visible operations for calculations or if you want to use all operations.

3.5.8 Send NC program file to machine

For some CNC milling machines this option can be used to directly send the NC program to the machine from DeskProto. **Most machines do not support this**, so by default this option is not available (grayed out). Also in many cases the machine will be connected to a different computer. The option will become available after configuring an output device (or program) to be used in the DeskProto Preferences:

In the Preferences (Options menu) a port like COM1: or LPT1: can be configured as the output device, or a Windows Printer driver. DeskProto will just copy the contents of your NC program file to the selected port or driver. Also see the paragraph on the DeskProto Preferences in this manual.
Note: this is possible for a few machines only, like the Roland CAMM-3 series.

It is also possible to configure an external program to send the NC toolpath file to. DeskProto will start this program, with the name of the NC-program file to be used as a command line parameter. The idea is to configure the control software of your CNC milling machine here. However in fact you can select any program, also for instance a milling simulation software, or (for the diehards) a plain text editor like Notepad to change the NC-program that DeskProto has just created.



In DeskProto the NC program file must be saved first; a drawn toolpath is not sufficient to use this option. A dialog as shown above will be displayed to ask you for the file you want to transfer. After selecting the correct NC program file and 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.

For Roland machines the above dialog will also remind you that the green LED for "View" needs to be off before sending.

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

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

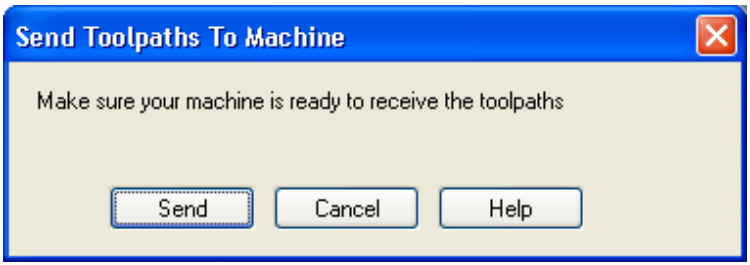
1. Fixture a fitting piece of material on the machine (look in the the Geometry Information dialog for the correct dimensions).
2. Mount the correct tool. Note: in case you use a different cutter than entered in DeskProto an incorrect part will be produced.
3. Tell the machine where to find the material.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

3.5.9 Send current toolpaths to machine

For some CNC milling machines this option can be used to directly send the NC toolpaths to the machine from DeskProto. **Most machines do not support this**, so by default this option is not available (grayed out). Also in many cases the machine will be connected to a different computer. The option will become available after configuring an output device (or program) to be used in the DeskProto Preferences:

In the Preferences (Options menu) a port like COM1: or LPT1: can be configured as the output device, or a Windows Printer driver. DeskProto will just copy the contents of your NC program file to the selected port or driver. Also see the paragraph on the DeskProto Preferences in this manual.
Note: this is possible for a few machines only, like the Roland CAMM-3 series.

It is also possible to configure an external program to send the toolpaths to. DeskProto will then save the toolpaths to a file called TempNC.ext, and then start the external program with the name TempNC.ext as a command line parameter. The idea is to configure the control software of your CNC milling machine here. However in fact you can select any program, also for instance a milling simulation software, or (for the diehards) a plain text editor like Notepad to change the NC-program that DeskProto has just created.



A dialog as shown above will be displayed to ask you if the machine is ready. After pressing Send DeskProto will start sending.
For Roland machines the above dialog will also remind you that the green LED for "View" needs to be off before sending.

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

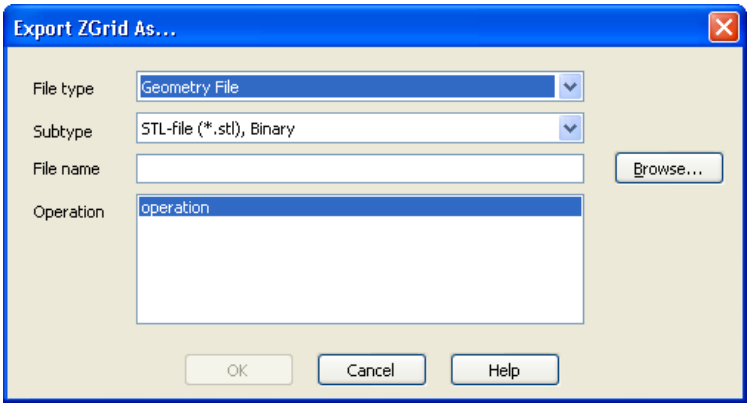
How to prepare the machine will be different for every machine. Still some general guidelines can be given. Basically the following steps have to be taken:

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

3.5.10 Export Z-grid 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 subjects 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 the Simulation as a geometry file. For standard DeskProto use this is absolutely not needed, however it might be useful for instance to use external software to compare original geometry and simulation.



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

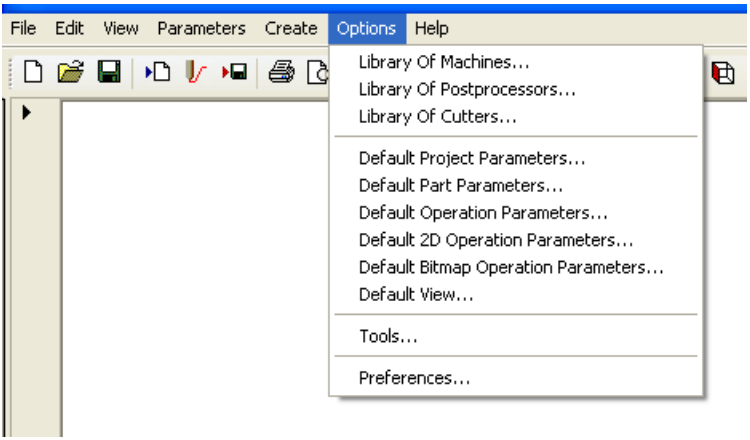
The **File type** is either Bitmap file, Geometry file or XYZ file.
The use of a geometry file was described above.
For export as Bitmap file DeskProto will convert the 3D information to 2D by translating Z-height to gray-value. For each point in the Z-grid one pixel will be made. The highest Z-value will be given a white pixel, the lowest level a black pixel, and all in-between values an according in-between grey 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: GIF, JPG, PNG and BMP.
For Geometry files you can choose any of the formats that DeskProto supports for 3D, as described in 3D Geometry.
For XYZ files only one format is supported: ASCII text file, one point with three coordinates per line.

The function of **File name** will be clear, you can either enter it in the edit box, to be save in the current directory, or browse the file using the button.

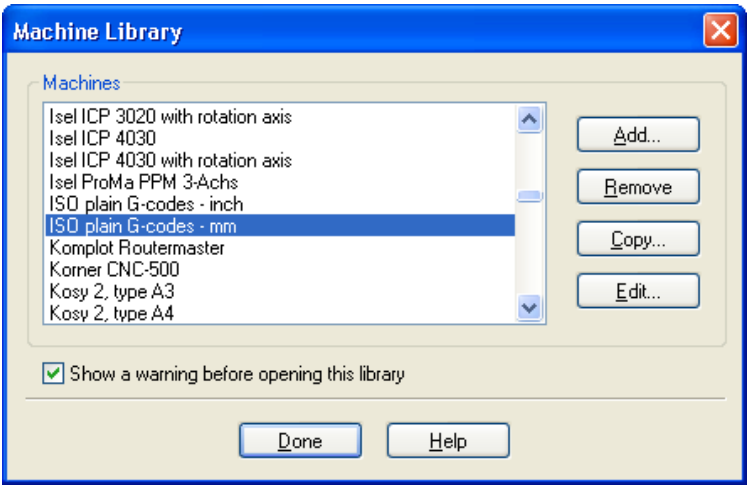
In the **Operation** field you can select which of the operations will be used. As each operation has one Z-grid and one Simulation, you can only select one operation here.

3.6 Options Menu



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

3.6.1 Library of Machines



For every NC program to be created you will need to select the machine-definition for the NC milling machine that you are going to use. A number of predefined machine-definitions have been included with DeskProto. This is the Library of machines, which has been copied to your computer during the automatic installation procedure of DeskProto. In most cases you can just select one of the existing machines when defining the Part parameters. However in case you have a special machine, you can edit an existing machine definition or define your own machine in the Machine library.

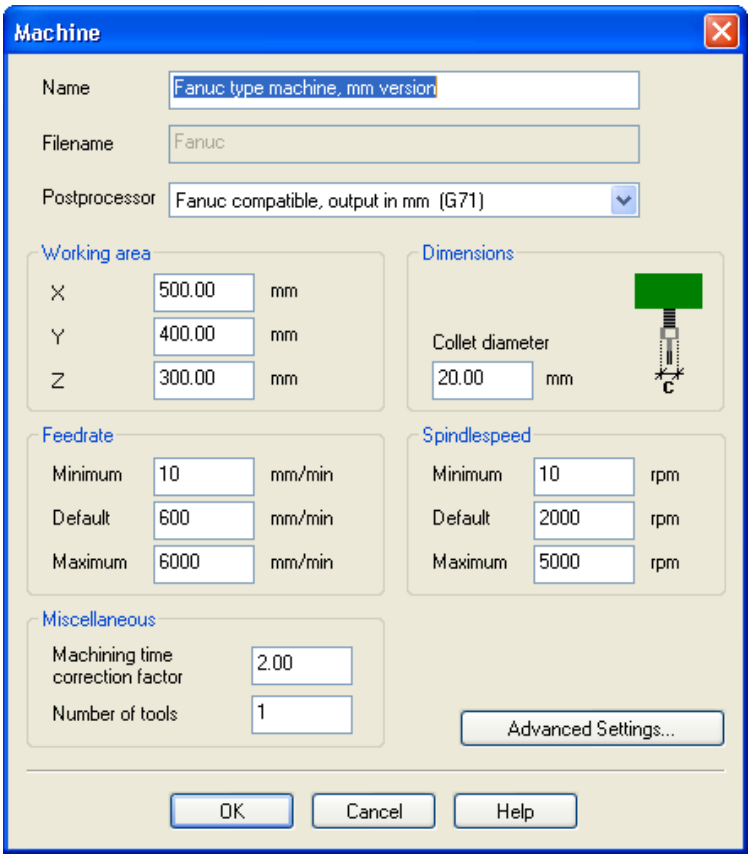
After a warning message that this option is meant for advanced users only, the dialog shown above will pop up. Here you can choose the machine you want to **Edit** or **Copy**, and also **Add** and **Remove** machines. After pressing Add, Copy or Edit the Machine dialog will be shown, containing all parameters to define a machine in DeskProto.
The warning message by default is enabled, you can disable it by un-checking the checkbox **Show a warning before opening this library**.

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

A machine definition is stored as a file name .mch in the DeskProto drivers directory (See Preferences). These files can be copied, for instance to a different PC to make the machine available on that PC as well. The files are in Windows .ini format, and can be changed using a plain editor like Notepad (changing in DeskProto is safer though). When changing in an editor, it is possible to add comments, for instance on the file history. Any line that starts with a semicolon (;) is a comment.

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

3.6.2 Machine dialog



The **Name** is the name that will appear in any DeskProto dialog for selecting a machine. It needs not be the same as the filename: use a name that clearly indicates which machine you mean. Each machine must have a unique name.

The **Filename** will be used to store the machine definition, using the file extension .MCH . When editing an existing machine you can no longer change the Filename. You can also add and remove machines by adding and removing MCH files to/from the DeskProto Drivers directory as set in the Preferences.

The **Postprocessor** you select for this machine will be used to make the actual NC programs: see the Postprocessor library.

Most other values you enter in the Machine dialog are in fact less important as they will be used only to check whether the parameters entered later do not exceed the machines' capabilities and do not influence the resulting toolpath (only the collet diameter does).

The **Working area** is used for validation, to see if all the toolpaths will fit in the reach of the machine. Also, when drawing the working area of the machine, a box of these dimensions is drawn.
The **Collet diameter** as defined at the Dimensions section is used for the Collet collision check: to prevent the collet from damaging the model.

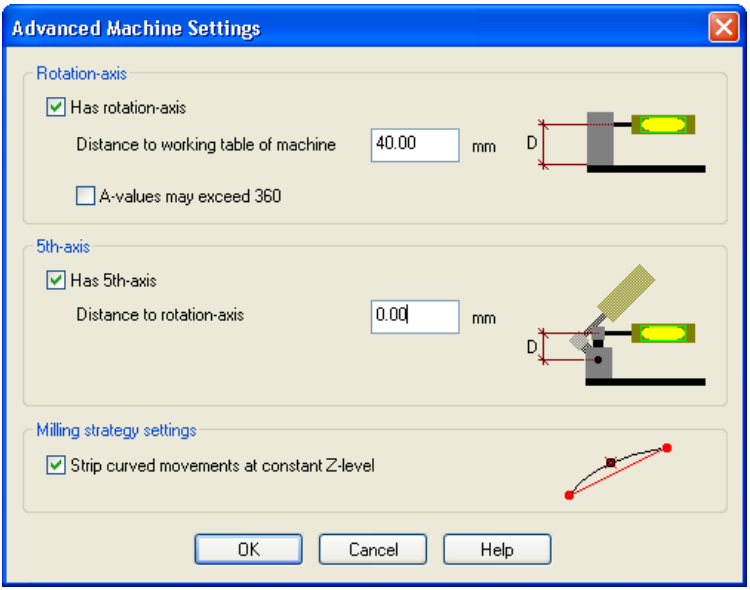
The **Feedrate** and **Spindle speed** Min and Max values set here are used to validate the speed settings in the operation parameters. The default values are used when selecting a different machine for a part, to reset the speed values for all its operations. In case your milling machine requires a feedrate command for rapid movements, that feedrate is determined by the maximum feedrate that is set here.

The **Machining time correction factor** is exactly what its name suggests: a factor that is used for this machine to multiply the theoretical machining time with in order to get an estimated real Machining time. This factor has to be a value larger than 1.
The **Number of tool**s is the number of tools that this machine can store and select automatically. Only machines that have an automatic toolchanger should use this. This option is also used for validation only.

The **Advanced settings** button leads to the Advanced settings.

3.6.3 Advanced machine settings

This dialog is a part of the Machine definition of DeskProto, and can be reached via the Advanced Settings button on the Machine definition dialog. The advanced settings configure the availability and dimensions of the optional rotation axis.



The option to indicate the presence of a **Rotation axis** has 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.
The rotation axis is a device that rotates the part during machining, meaning that you can machine around the part. Image it like a peace of meat rotating above a barbecue. This is a very common option on CNC milling machines.
In DeskProto only an A-axis is supported, which means that the axis of rotation is parallel to the X-axis of the machine.

The **Distance to working table of machine** is the distance between the actual rotation axis center line and the machine table below: this value determines the maximum block diameter (or in fact radius) that still can be rotated on this machine.

On most 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. Some machines allow you to keep rotating in one direction: you then can use the option **A-values may exceed 360** degrees. Note that the angle value that is sent to the machine will then keep growing (for instance after 100 rotations it will be A = 36000 degrees).

The **5th axis** as supported by DeskProto is absolutely not common: only a few machines support this. As the icon on the dialog shows the complete 4th axis unit will be tilted, like a draw-bridge being opened. As this rotation is

around the Y-axis, technically speaking this is a B-axis rotation. The advantage is the when machining a ring this rotation allows you to also machine the inside of the ring. Note that DeskProto only supports **manually controlled** 5th axis.
For a picture see the 5th axis tab of the Operation parameters.

If this option is not checked, the option "Use 5th-axis" in the Part parameters will be grayed out.

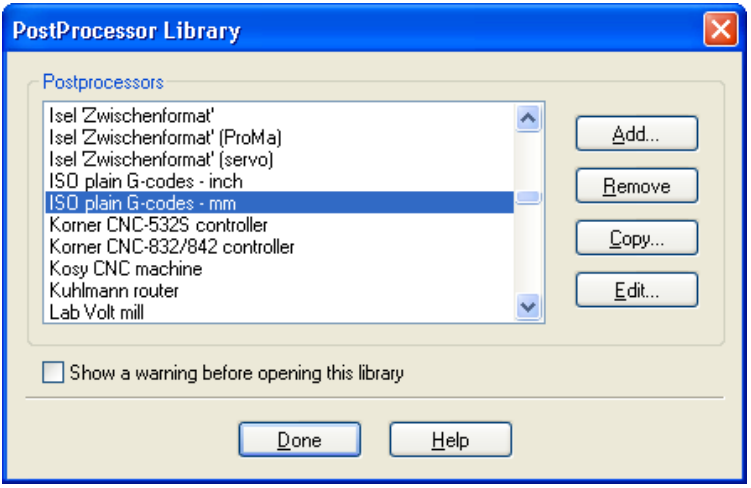
Besides the checkbox only one parameter is available: the **distance to rotation axis** which sets a vertical distance between the A-axis and the B-axis. Note that for most machines with a 5th axis (like the Roland JWX-10 and MDX-40) this distance is 0.00

The third advance machine setting is an optimization for the toolpaths: **Strip curved movements at constant Z-level**. This optimization makes the NC file shorter by deleting some intermediate points on the toolpath.
Each toolpath is built using a large number of short straight lines, the length on each line-segment determined by the Stepsize along toolpath. A long straight toolpath will also have been calculated as a series of segments, and for such straight movement in fact all intermediate points can be skipped: one long line will result in the same cutter movement. This optimization for straight lines is done automatically in DeskProto.
This easy approach is not possible for curved toolpaths like Circular and Spiral, as DeskProto does not support arc movements. A circular toolpath also is built using a large number of short straight lines, however here deleting a point slightly changes the toolpath. Still when machining a flat horizontal surface such small changes do not matter at all.

For some machines deleting points will make the movement faster and also smoother, as the controller of that machine does not have enough calculation power to calculate each small movement in time to keep the machine on speed. The result will be a non-smooth movement. For such machine you may check this option, then DeskProto will delete half of the points on the circular toolpath. The result will be a faster and smoother movement of the cutter.

Note that for other machines checking this option will have a contrary result and make the movement slower. This will happen for machines with a very fast controller, that checks the angle between two consecutive line segments on the toolpath and only keeps up speed in case almost parallel. Deleting intermediate point will increase the angle between the remaining line segments, so for these machines you should not check this option.

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

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

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

The warning message by default is enabled, you can disable it by un-checking the checkbox **Show a warning before opening this library**.

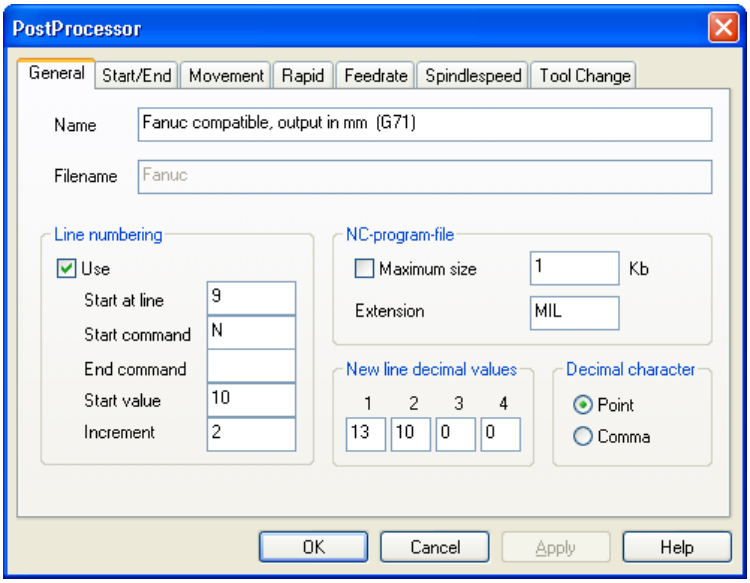
A postprocessor definition is stored as a file name .ppr in the DeskProto drivers directory (see Preferences). These files can be copied, for instance to a different PC to make the postprocessor available on that PC as well. The files are in Windows .ini format, and can be changed using a plain editor like Notepad (changing in DeskProto is safer though). When changing in an editor, it is possible to add comments, for instance on the file history. Any line that starts with a semicolon (;) is a comment.

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

3.6.5 Postprocessor dialog

The postprocessor dialog is divided into 7 tab pages.

General settings



The **Name** is the name that will appear in any DeskProto dialog for selecting a postprocessor. It needs not be the same as the filename: use a name that clearly indicates which postprocessor you mean. Each postprocessor must have a unique name.

The **Filename** will be used to store the postprocessor definition, using the file extension .PPR . When editing an existing postprocessor you can no longer change the Filename. You can also add and remove postprocessors by adding and removing PPR files to/from the DeskProto Drivers directory as set in the Preferences.

When the output file needs **Line-numbering**, switch on the 'use' button in the line numbering group. There you can also define at which line in the file the numbering should start ('start line'), with which commands the numbers should 'start' and 'end', the 'start value' and the 'increment'.

In **NC Program File**, setting a **Maximum size** is needed for some older types of 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** will be used for every NC program file that will be created using this postprocessor. It does not influence the contents of the file, just the file's name.

The values you enter at the **New line decimal values** group will be put behind every line of the output file. Default these are set to the values 13,10,0,0 which will do for almost any machine. The value 13 stands for carriage-return, the value 10 stands for line-feed, and the values which are 0 will not be used by the postprocessor (unless followed by a non-zero value). Do not touch these values unless you are absolutely sure what your do !

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 as either 3.50 or 3,50

Start / End Settings

This page contains two editing windows called **Start commands** and **End commands**. Here you can enter the lines that every NC-program must start with, and the lines it must end with. This may include things like turning on the spindle motor, setting the units to metric or imperial, and other global

functions. Please look at an existing postprocessor for an example. Also looking in an existing NC program file that works OK on your machine is helpful here.

Movement settings

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

Each movement (so each line in the NC file) is built by a Start command, some Coordinates and an End command. To see their effect: just look at the Example line to see what will happen. The **Start command** determines which motion type is called for, the **Coordinates** where to. Most machines do not require an **End command**.
The option **Only for first movement** makes the Movement command global: 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) may be configured separately:
The **Start command** and **End command** per coordinate determine if the coordinate is for X, Y, Z or A.
Positions gives the minimum number of character positions to be used. So if units is set to 7 and the output is 3.000 two spaces will be added, making the result in this example 'X 3.000'
Decimals gives the number of units to be written behind the decimal point (or comma), so this affects the precision of the output.
The output of DeskProto is either in mm or inches (whichever you configured). When your output should be in any other unit (like 1/100 mm) you can change the Factor for the X, Y, Z coordinates.
Write only if changed means that coordinate values are only written in case changed. So the command "G1 Y20.0" means 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.
Using the **Order** combo box you change the order of the X, Y, Z and A coordinates in the output.
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 also see the example line.

In the **Units** group you can choose between mm and inches for the coordinate values to be converted to. Make sure that your machine uses the same units: for some machines you need to explicitly give the command G 70 (inches) or G71 (mm). You can do so at the "Start commands' of the Start/End tab page. Do also make sure that you set the DeskProto preferences

to the correct Units. The **Angle** units are only used for rotation axis machining, so do not apply if your machine does not have a rotation axis.

Do note that the (A) column and the Angle units are only used in case a Rotation axis is both present and selected. For three axis machines you can just ignore the values entered here.

Rapid Settings

Rapid movements are used to save milling time by moving as fast as the machine can travel. These are used 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 global: after been given once it stays valid until a different command is given.

Feedrate settings:

Feedrate commands will only be output if the option **Use** is switched on.
The option **Write only if changed** will be on for most machines: if not the feedrate will be output on every movement line.
The effect of using the **Start-command** and **End-command** can best be seen by looking at the example line at the bottom of this tab page.
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).
The **Method** setting is self-explanatory.
The settings of **Units** are only used to be able to show the correct units in the dialogs, whenever you have to enter a feedrate. DeskProto does not "understand" them, it just writes the number to the NC program file.

Note that an invisible special option is available in the fields Start command and End command: you can enter new lines when needed to create a multi-line Feedrate command. Entering the string “^\N” will start a new line, and entering the string “^\V” will again output the actual feedrate value.

For Instance: entering the string “ F ^\V^\N G” in the Start command edit box will cause DeskProto to write a two-line Feedrate command as follows:
F 500

G 500
(given that the actual Feedrate value to be output is 500).

Spindle speed settings

Spindle commands will only be output if the option **Use** is switched on.
The option **Write only if changed** will be on for most machines: if not the spindle speed will be output on every movement line.
The effect of using the **Start-command** and **End-command** can best be seen by looking at the example line at the bottom of this tab page.
Positions stands for the minimum number of positions that are occupied by the value of the spindlespeed.
Decimals stands for the number of characters behind the point (or comma).
The **Method** setting is self-explanatory.

Note that an invisible special option is available in the fields Start command and End command: you can enter new lines when needed to create a multi-line Spindle speed command. Entering the string "^\N" will start a new line, and entering the string "^\V" will again output the actual feedrate value.

For Instance: entering the string " S ^\V^\N R" in the Start command edit box will cause DeskProto to write a two-line Spindle speed command as follows:
S 8000
R 8000
Given that the actual Spindle speed value to be output is 8000.

Toolchange settings

DeskProto will create one NC program file for one part. However, for every operation a different cutter can be used, so in case you are using different cutters in one part (which would mean in one NC program) you have to define what should happen at a tool change. Just choose 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 (ATC = Automatic Tool Change). 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. See the Example line on the bottom of this dialog for an example. In the illustration above note the space just before

the “M06”. This is just an example command: actual commands will differ per machine.
The same special options ^\V and ^\N as just described for Feedrate are available here as well, so if needed you can use more than three lines.

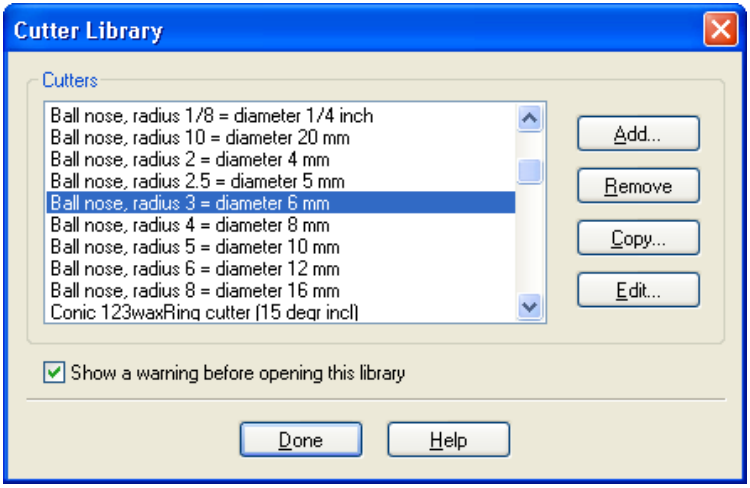
Note:
For this method be sure that the ‘number of tools’ of the machine is set correctly. You can do that at the Machine dialog. Also be sure that the cutters that you use in DeskProto have the correct ‘number in machine’ parameter. You can set that at the Cutter dialog. You also have to check if the correct cutter is indeed loaded on this position of the machine’s tool changer: DeskProto just loads Tool No “N” without knowing if it is indeed the right cutter.

Use pause-command in NC program lets you change the cutter manually, however within the same NC program. The Pause command has to 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.

Use new NC program will cause a new NC program to be started. If your machine does not support a tool change or if you do not know, it is best to choose new NC program file. After ending the first file you can then change the cutter, correct the zero position for the length of the new cutter, and start the second NC program file.
Names for the subsequent files will be automatically generated. In case you chose the name Test.nc and the two operations were called Roughing and Finishing, then file first file will be called Test#1_Roughing.nc and the second file Test#2_Finishing.nc

Note:
Use new NC program is the default option and also the safest.

3.6.6 Library of Cutters



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

After a warning message that this option is meant for advanced users only, 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.

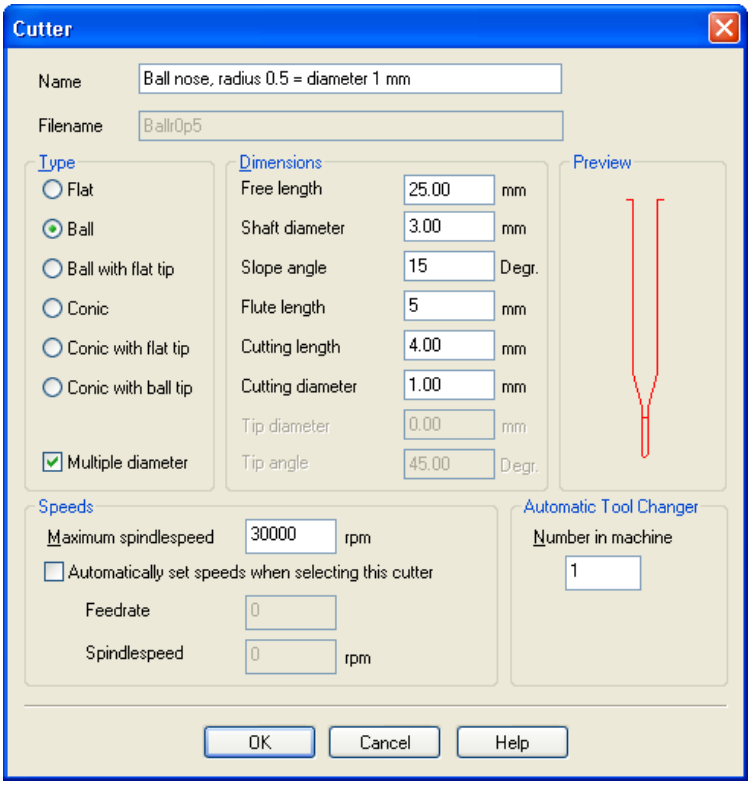
The warning message by default is enabled, you can disable it by un-checking the checkbox **Show a warning before opening this library**.

A cutter definition is stored as a file name .ctr in the DeskProto drivers directory (see Preferences). These files can be copied, for instance to a different PC to make the cutter available on that PC as well. The files are in Windows .ini format, and can be changed using a plain editor like Notepad (changing in DeskProto is safer though). When changing in an editor, it is possible to add comments, for instance on the file history. Any line that starts

with a semicolon (;) is a comment.

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

3.6.7 Cutter dialog

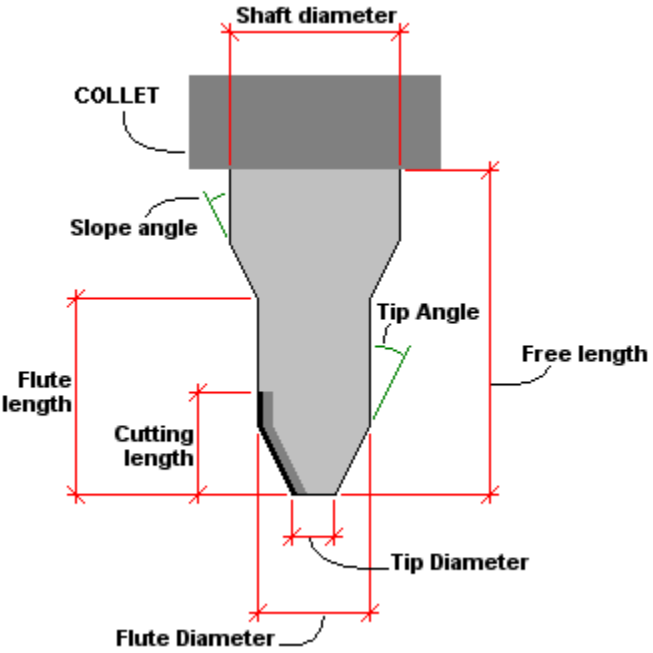


The **Name** is the name that will appear in any DeskProto dialog for selecting a cutter. It needs not be the same as the filename: use a name that clearly indicates which cutter you mean. Do show whether you mean radius or diameter for any number in this Name (you will forget if 'Ball6' means R6 or D6). Each cutter must have a unique name.
The **Filename** will be used to store the cutter definition, using the file extension .CTR . When editing an existing cutter you can no longer change the Filename. You can also add and remove cutters by adding and removing

CTR files to/from the DeskProto Drivers directory as set in the Preferences.

DeskProto offers six **Type**s of cutter, that are variants of three basic types: flat (square cutter), conic (sharp point), and ball (ballnose cutter). Both conic and ball can be given a flat tool-tip, conic can be given a ballnose tooltip as well. Depending on the type you choose some dimensions might be disabled. For example a flat cutter does not have a tip-angle.
For each type you can also 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 very standard model for small cutters. When correctly defined, DeskProto will make sure that the thick shaft does not damage your model at vertical surfaces.

Just look at the drawing in the **Preview** to see the cutter that you have defined.
The image below gives an explanation of most terms used for the **Dimensions**.



The **Free length** is the length of the part of the tool below the collet. So this is not the total length of the cutter. The free length is not constant, as it depends on how far you insert the cutter in the collet. DeskProto uses this

parameter only for the collet collision check.
The meaning of **Shaft diameter** will be clear. It is only available for multiple diameter tools, as otherwise it is the same as the cutting diameter.
The **Slope Angle** also is available only for multiple diameter cutters.It defines the transition between the thick shaft and the thinner flute, which for most cutter has a conical shape.

The **Flute length** is only available for multiple diameter tools, as it is the length of the small diameter part which is called the flute of the cutter..
The **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 go deeper than this value in a single pass. In case you do not select Roughing then DeskProto will automatically do so for the first Operation.
The **Cutting diameter** (or flute diameter) is the nominal diameter of the cutter, used for the calculations (do not confuse with Radius).

The **Tip diameter** is the diameter of the flat tip for cutter types with a flat tip, and the diameter of the ballnose tip for Ball/Conic cutters. So for instance a Ball/Flat cutter with diameter 6 and tip 4 will have a radius 1 left (this type of cutter sometimes is called bull nose).
A different cutter type can be defined by choosing “Ball with flat tip” and setting the Tip diameter larger than the Shaft diameter (which normally is nonsense). This creates a special cutter type “Curved tip” as a result. The preview drawing shows what will happen.

The **Tip angle** is for conic cutters only: it is the angle between the cutting edge and the center-line of the tool (so the ‘grinding angle’, not the ‘ included angle’). In case a parameter is not yet clear, just try and look what happens in the Preview. You can also use this parameter to define special tapered cutters having a draft angle of say 3 degrees.

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 correct dimensions have been entered first. The dashed horizontal line shows the cutting length, the two horizontal lines at top indicate the collet and show the free length.

The **Number in machine** indicates on which position of the Automatic Toolchanger this cutter needs to be loaded. So this is useful only when you are using a machine that supports automatic toolchange. The number will be used for any toolchange command written in the NC-program file.
Important:
This number can be different for each machine and even for each situation, as the operator has to load the correct cutter in the correct location.

The **Maximum spindlespeed** is used only for validation of your projects (some larger cutters may not rotate at high rpm as these are insufficiently balanced).

The option **Automatically set speeds when selecting this cutter** may be very handy when you have standard milling conditions. For instance when you always machine in wax (jewelry wax models) you may want to always use the same low feedrate and high spindle speed for a certain thin cutter. Then you can define these two speeds here, and in the Operation parameters DeskProto will automatically set these values for **Feedrate** and **Spindlespeed** when this cutter is selected.

Note that for the Feedrate DeskProto will just copy the number that you enter in this dialog to the NC file, without understanding it's units. So when enter "10" and use this cutter on a m/min machine the feedrate will be 10 m/min, on a mm/sec machine it will be 10 mm/sec. A large difference !!! DeskProto could show the units used by your current machine, however that would give a false sense of security as they would be valid for that machine only.

3.6.8 Default project

After choosing this option, first you will be warned that whatever you change here will influence all new projects that you create later. The dialog that then 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 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 Flip normals and Skip backfaces are allowed, which are very useful as default in case your CAD system always produces incorrect STL files. The DeskProto default is that both Flip Normals and Skip Backfaces are 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 specification is not possible as that would conflict with the default Part and the default Operations.
Note that can also achieve much using command line parameters, by auto-loading a special project file, which may have more Parts. For instance for automation of two sided machining.

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

3.6.9 Default part

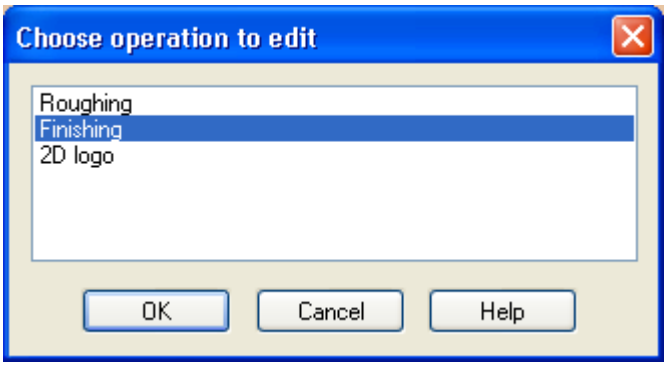
After choosing this option, first you will be warned that whatever you change here will influence all new parts that you create later.
The dialog that then pops up is equal to the Part Parameters dialog, it will thus not be explained here. The difference is that this function adjusts the default part settings, that will be used for every new Part. This is the option to use for instance to have your milling machine automatically selected, when you want to use more than one operation for all your parts, when you have a machine that needs a specific translation method, etc, etc.

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

3.6.10 Default Operation (3D)

After choosing this option, first you will be warned that whatever you change here will influence all new operations that you create later. The dialog that then pops up is equal to the Operation Parameters dialog, it will thus not be explained here. The difference is that this function adjusts the default operation settings, that will be used for every new Operation. This is the option to use for instance in case you want a specific cutter automatically selected, need a specific feedrate or spindle speed, etc, etc.

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.

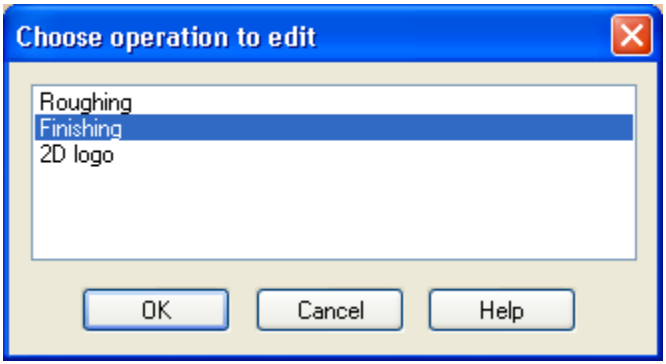


These default parameters will be stored in the Windows registry, and will be different for every user. As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button is supplied here to restore the factory default settings for all parameters.

3.6.11 Default 2D Operation

After choosing this option, first you will be warned that whatever you change here will influence all new 2D operations that you create later. The dialog that then pops up is equal to the 2D Operation Parameters dialog, it will thus not be explained here. The difference is that this function adjusts the default 2D operation settings, that will be used for every new 2D 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. It is not possible to set a default 2D file.

In case more than one default 2D Operation has been defined in the default part, first a dialog is displayed in which you can select one of the default operations.



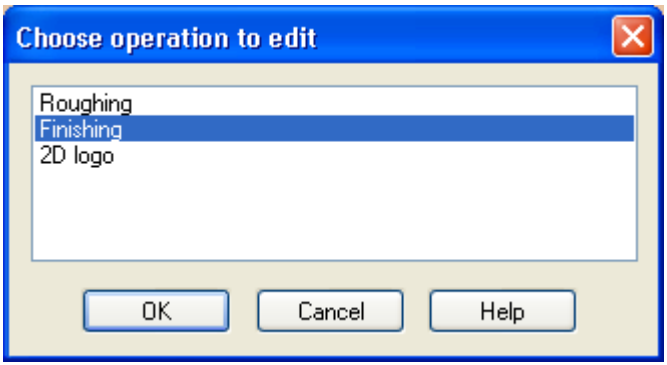
These default parameters will be stored in the Windows registry, and will be different for every user. As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button is supplied here to restore the factory default settings for all parameters.

3.6.12 Default Bitmap Operation

After choosing this option, first you will be warned that whatever you change here will influence all new Bitmap operations that you create later.

The 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 settings, 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. It is not possible to set a default Bitmap file.

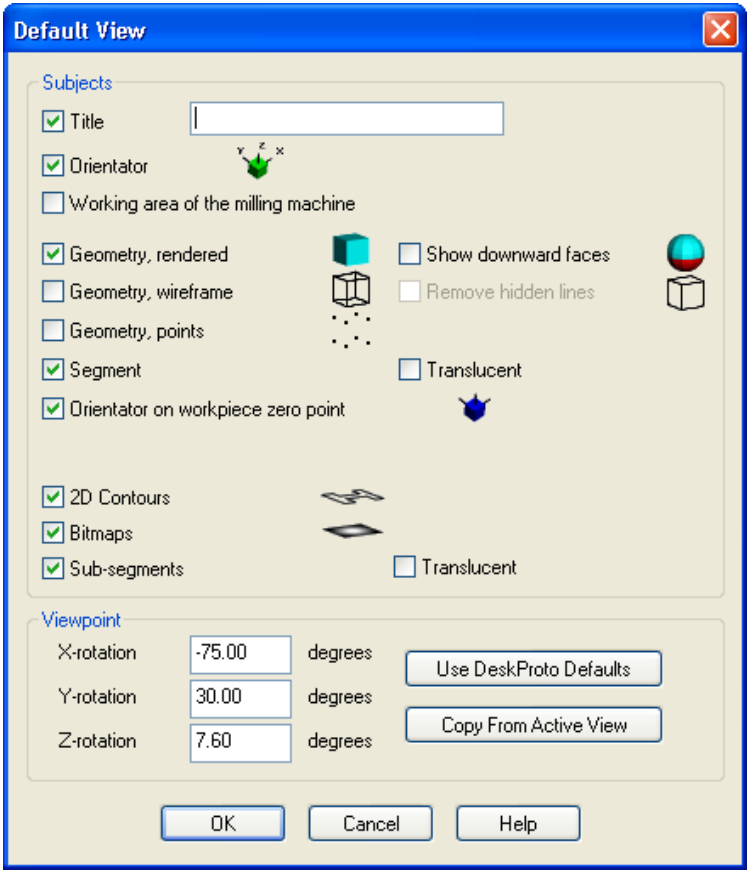
In case more than one default Bitmap Operation has been defined in the default part, first a dialog is displayed in which you can select one of the default operations.



These default parameters will be stored in the Windows registry, and will be different for every user. As it is possible to completely mess up these settings, making it very hard to use DeskProto, an extra button is supplied here to **Restore the factory default settings** for all parameters.

At this location you can also call the dialog to change the Default Bitmap Settings.

3.6.13 Default View dialog



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

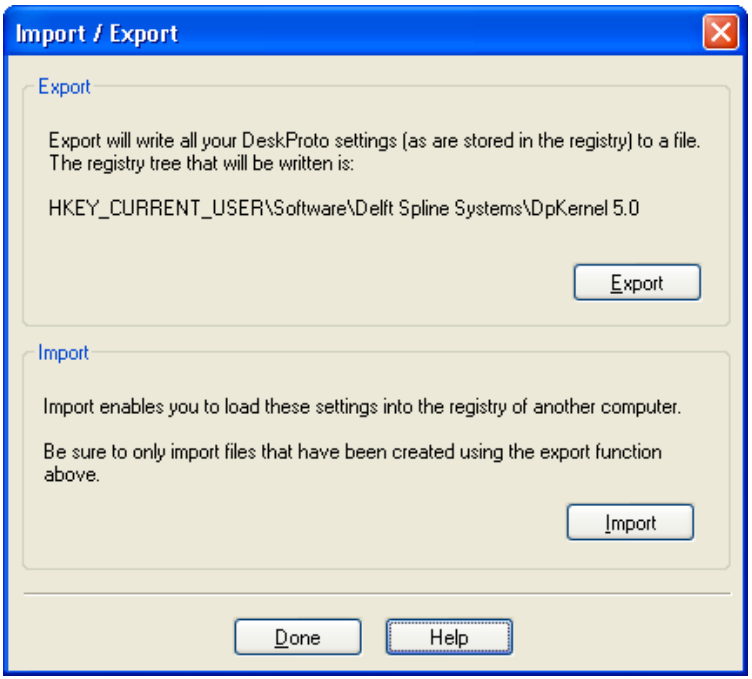
To edit the parameters of the default view a special dialog is used, as both the dialogs to change the Subjects and the Viewpoint contain parameters that are not suitable as default parameters.

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

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

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

3.6.14 Tools dialog



This is a dialog that offers extra **Tools**.
Currently only one Tool is available: importing and exporting the **Registry** settings.

All DeskProto defaults and preferences are stored in the Registry, conform the specifications for Windows applications as made by Microsoft. This use of the Registry 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.

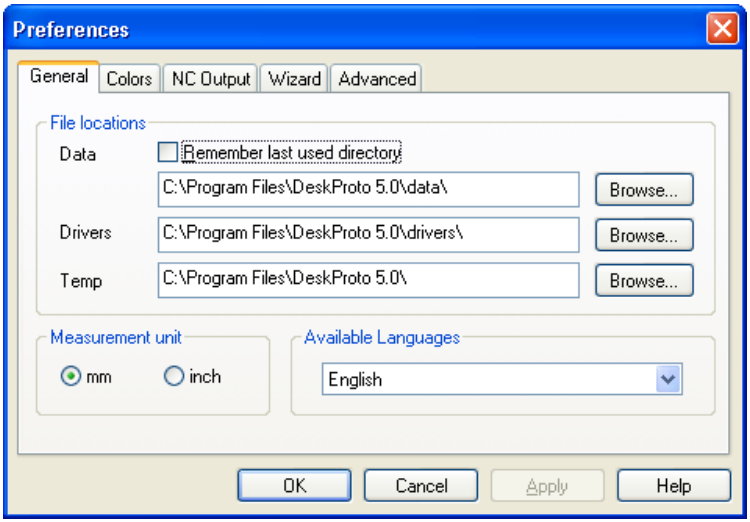
This is where the **Import** / **Export** tool comes in: this tool allows you to export DeskProto's Registry settings to a file, and also to import all these settings from a file that has been created previously.
The file format used will be a .reg file.
Note that such file can contain many other registry settings as well: when importing DeskProto does not check this. So make sure that any file that you import here has indeed been created by DeskProto.

The use of this dialog is straightforward: button Export will export a .reg file, button Import will import one.

3.6.15 Preferences dialog

This is a dialog in which you can edit some application preferences. The dialog consists of 5 Tab pages.
These preferences are stored in the registry. Each user has its own preferences stored there.

General tab page

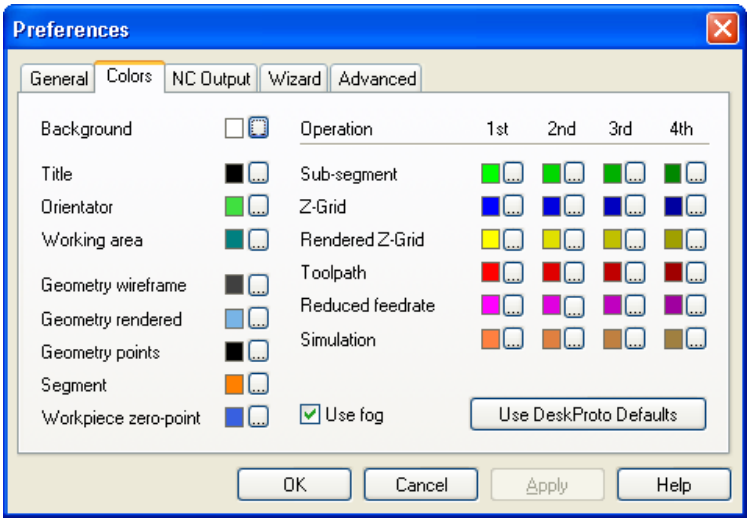


The **File-locations** listed here are the default directories where DeskProto writes and looks for the specified file types. These directories can be customized in this dialog by clicking at the appropriate field and then typing the new directory's complete path-specification(i.e.: C:\DeskProto\ ), or by using the Browse button.
Data is the location where DeskProto will initiate the Load and Save dialog boxes. The Data directory is the only one having the extra option **Remember last used directory**, making DeskProto remember which directory you were working in the last time you used DeskProto.
Drivers is the location from which DeskProto will load the drivers (machines, postprocessors and cutters) while starting up the application. Any valid new files that you copy to this directory will be automatically available after starting DeskProto. Changing this location removes all drivers from the library (not from disk though) and loads drivers from the new location (if available). Because the open project uses drivers from that library, the open project will be closed before removing the library. A new empty project will be created after the new library has been loaded or created at the new location.
Temp is the location where DeskProto stores temporary files which will be removed again when closing DeskProto.

Using the **Measurement unit** you can choose between metric (mm) and imperial (inches) for your DeskProto configuration. As the STL files that you use do not state the units used, DeskProto has to assume that they are in the same unit as the one you define here.
Note: also check your milling machine's documentation to see which units are needed for the machine, as that setting is independent from this preferences setting. Some machines expect NC files in mm, some will expect files to be in inches. Make sure to configure your postprocessor to use the correct units: this is completely independent of the choice that you make in these Preferences. DeskProto will then automatically perform the unit conversion in case this is needed.

Other **Available languages** than English are only present in case you have copied the appropriate files for a certain language to the Lang subdirectory of the DeskProto installation directory (for instance C:\Program Files\DeskProto 4.1\Lang).
For each language at least the following two files are needed (containing the translated resources for Deskproto.exe and Dpkernel.dll): *ResourceExeName.dll* and *ResourceKernelName.dll* in which "name" stands for the language. For instance "Deutsch" for German needs files *ResourceExeDeutsch.dll* and *ResourceKernelDeutsch.dll*
In addition translated Help files are needed.
Do contact your local dealer to see if a translated version for your language is available.

Colors tab page

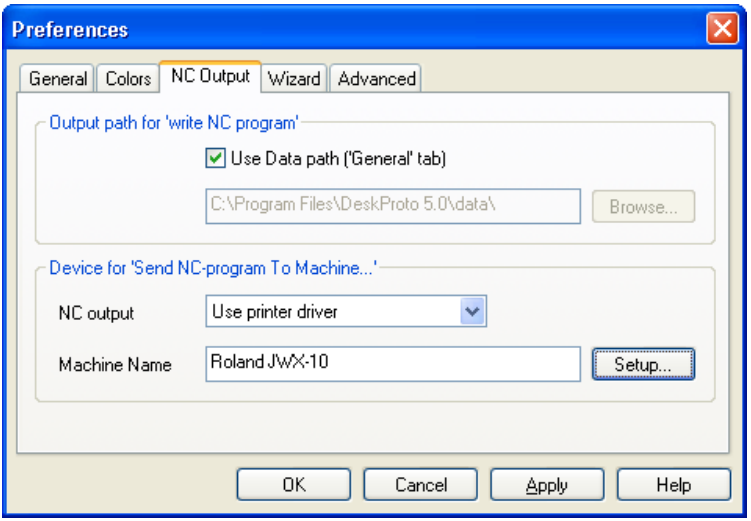


The tab page Colors makes it possible to customize all colors used on the DeskProto screen: just use a **[...] button** to change any color shown. As you can see different operations can be assigned different colors. In case of many operations: the color for operation No. 1 will also be used for No. 5, 9, 13 etc etc.

Checking the option **Use fog** applies fog to all drawings using lines. This technique, also called ' depth cueing', improves the perception of depth on your two-dimensional screen by making lines more vague as the distance is larger (as if obscured by a light fog).
On Black and White printers this may cause the lines to be printed like dotted lines, resulting in a blurry print. Therefore you might want to temporarily switch off the fog when printing. On computers with a video-card which does not support OpenGL this option might not have effect.

In case you have made a giant mess of your color settings you can press **Use DeskProto defaults** to restore the default settings of DeskProto for all colors (which are stored internally in DeskProto and which can not be changed).

NC Output tab page



These preferences are meant to configure the NC-output, so the exporting of NC files by DeskProto.

With the first preference you can fine-tune the output path. Normally DeskProto exports the NC files to the same directory from where the project and/or the geometry file were opened. 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 directory of the PC next to the milling machine).

The second preference refers to the commands Send NC-program to Machine and Send current toolpaths to Machine in the Create menu. Here you can make this command available or not, and determine which process has to be started after giving the command and selecting the NC file to be output.

NC output
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:

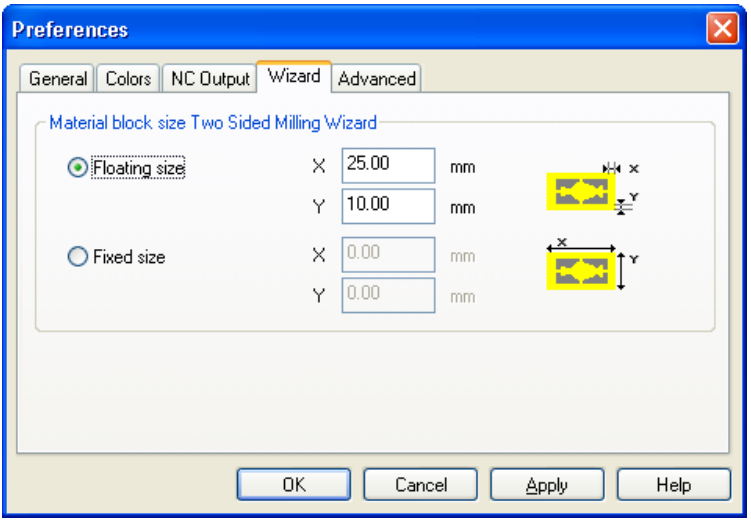
- In case you configure **None**, the command Send NC-program To Machine will not be available (grayed out). No sub-settings are available.

- Selecting a **Hardware Port** (either serial **COM..** or parallel **LPT..**) means that the NC-program will just be copied to that port after the command is given. This can be for example LPT1 or COM2. For a Centronics Printer port (LPT1 or LPT2) no settings are needed. In case of a COM-port you need to configure this port according to the specifications of the milling machine Use the button **Setup** for the standard Windows **COM port settings** dialog (setting values like 'Baud Rate', 'Data Bits', 'Parity', 'Stop Bits' & 'Flow Control').

- Selecting **"Use 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. Here you have to use the **Setup** button to select the correct printer driver. Take care not to select a 'normal' printer as this will results in hundreds of pages to be printed.
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 might be needed to select "Print directly to printer" in the Advanced Printer properties of the selected printer driver.

- After selecting the option **External program**, you can use the **Browse** button to define which program has to be used. 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. However in fact you can select any program, also for instance a milling simulation software, or (for the diehards) a plain text editor like Notepad to change the NC-program that DeskProto has just created.

Wizard tab page

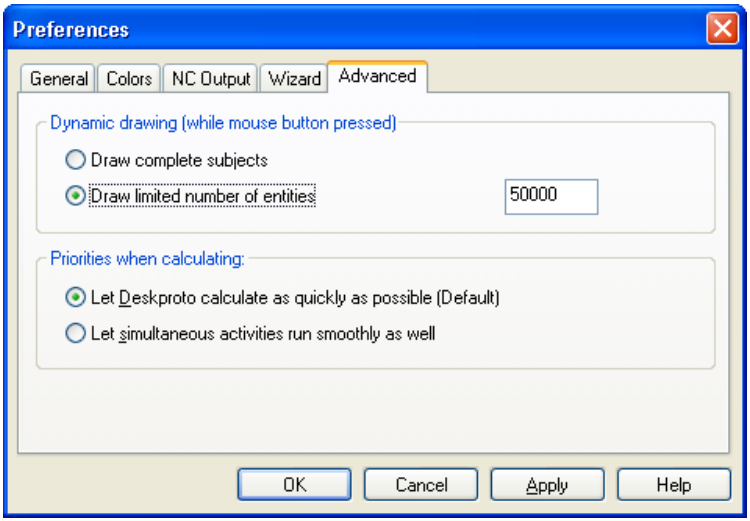


The tab page Wizard contains a preference that you can set for the Two-Sided Milling Wizard: the size of the material block. Two options are available:

- **Floating size**. The block size needed of course depends on the size of the part that you want to create. DeskProto offers a flexible solution here, called Floating size: around the part (including support blocks) a frame is added of a certain thickness. In the edit boxes for X and Y you can enter which frame thickness to use.
- **Fixed size** is useful in case all your parts have almost the same size: you then can use standard blocks of a certain size, to be defined in the X and Y edit box.

Only the X and Y block size can be set here, the Z dimension is set in the wizard.

Advanced tab page

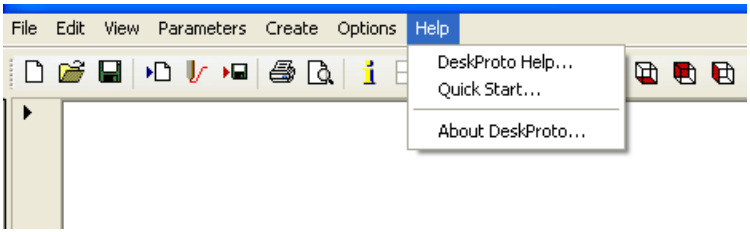


Dynamic drawing
Dynamic drawing is the drawing that is done while the left mouse button is pressed inside the view window or while using the thumb-wheels. Choosing the option **Draw limited number of entities** and decreasing the number shown there, will increase the speed with which you can move the geometry and other subjects (the term entities used here stands for points, lines and facets). If several subjects are drawn the maximum number that you enter is split up over all visible subjects. 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.

Priorities when calculating sets the allocation of the PC's resources.
DeskProto is a multithreaded application (Version 4.1 and higher). Calculating and drawing are done in different threads, as are of course other applications that run simultaneously. Standard, for heavy toolpath calculations DeskProto uses as much computer-power as it can get (calculations are done with a high priority). This may cause drawing and other applications to become slow and show jerking behavior. You can select **Let simultaneous applications runs smoothly as well** to give DeskProto's calculations a lower priority.
This functionality is no longer useful on newer dual-core processor PCs as

then while one core does the heavy calculations the other core still is available for all drawing activities.

3.7 Help Menu



Finally the Help menu which gives access to the Help system and to the About box.
For DeskProto Lite two extra options are present, Registration and Upgrading to DeskProto Full.

3.7.1 DeskProto Help

The Contents page of the DeskProto Help System, shown at the **Contents** tab on the left of the Help Window, offers an overview of all available Help topics, organized in 'books'. Double-click on a book to open it, and click on one of the items shown to see the appropriate Help page. In addition to the Contents tab page two more tabs are available:
the **Index** page can be used to find help on a specific subject, and
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 printed in blue and underlined, while the cursor will turn to a hand when moved over an active item. The DeskProto Help program also contains many active pictures (buttons for example), that can be clicked upon to get more information.

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

- The Help button on the Toolbar: 
- The button for Context Sensitive Help on the Toolbar: 
- Pressing the F1 Function key on the keyboard
- All dialogs also have a Help button, providing Help information about the use of that particular dialog.
- The Help system can also be started independently from DeskProto using

the Windows Start button (start menu option 'Programs', group 'DeskProto 4.0') and finally you can define a shortcut on your Desktop to access the DeskProto Help.

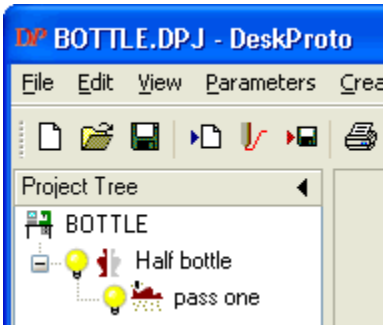
3.7.2 Quick Start

DeskProto is delivered with a printed Tutorial, meant to introduce you step by step to the functions that DeskProto offers. It is recommended to read **and** execute at least lessons number one and two of this Tutorial before starting to make prototypes with your own geometry.

However, in case you are no great manual reader and want to start at once exploring DeskProto, do read this Quick Start first. It is meant to explain the basic ideas of DeskProto, and you will need this information to be able to understand what is happening. After that, novel users are advised to first use the Wizard, that will guide them through all steps needed to generate an NC program file.

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

Within one project you can define parameters on three levels:
1. Project parameters are (amongst others) the name of the geometry file and the number of Parts that you want to use to create this prototype. Each of these Parts has its own set of parameters:
2. Part parameters define **what will be milled**. These set size, orientation, position and alike.
Within each Part you can use one or more milling Operations. Each Operation has its own set of parameters as well.
3. Operation parameters define **how it will be milled**.
These are in fact the only real 'milling parameters'. Note that three different types of operation are available: in addition to the standard (3D) Operation, a 2D Operation and a Bitmap Operation are available as well (not covered in this quick start).



You can imagine the tree-like structure of a project, which is displayed in the Project Tree at the left side of the DeskProto screen: see the figure above. The basic Project consists of one (nameless) Part and one (nameless) Operation. In the illustration the project is called Bottle, containing one Part called 'Half bottle' and one operation called 'operation one'.

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

The functions of DeskProto can be reached using the pull-down menus (or alternatively using the button-bar or the right mouse button). The most important menus are:

* The **View Menu** offers the opportunity to change the way you look at the geometry. Do also try to change your view by rotating the six coloured thumb wheels on the screen, and by using your mouse inside the view window. In fact most functionality in the View menu can be easier reached using the button bar.

* In the **Parameters Menu** you can edit all geometry- and milling parameters. For simple prototyping it is sufficient to edit only the front Tab screen for both Part and Operation: the other Tabs can come later as all parameters have suitable default values.
Do note that two different versions of DeskProto are available: DeskProto and DeskProto Lite: in the latter version only a subset of these parameters is available.

* Most important is the **Create Menu**, where you can start the milling calculations and write the NC program file.

We do hope you will enjoy using this software, it sure will help you to make

your Prototyping really Rapid.

3.7.3 Register

DeskProto Lite can be obtained by downloading the free 30 days lite-trial version from the Internet. This trial license is fully functional, however after the 30 days trial period the saving functions will be disabled. The trial license can be converted to a real license any moment (so both during the trial period and after) by entering a valid combination of Name and Registration code in this dialog.

A Registration code can be obtained using the order form that pops up at the button “Order form”.
In order to register please fill out this form, print it and fax it to Delft Spline Systems. You will receive your unlocking code as soon as possible. Do note that this code belongs to the Name that you entered in the Order form: both the Name and the Code have to be entered exactly as given, including case (upper of lower), spaces, punctuation and the like. Make sure to carefully store the Name and Code supplied, as you will again need it should you upgrade your computer.

After having registered, the registration dialog is no longer needed, and will be replaced by a Registration Information dialog.

3.7.4 Upgrade

Displays a dialog screen that lists the advantages of the full version of DeskProto over DeskProto Lite. The difference between both versions is the number of parameters that you can set (both Part parameters and Operation parameters). This dialog shows the most important extra parameters that will be available after upgrading your license. Of course for registered users of DeskProto Lite the full version is available at a special upgrade price.

3.7.5 About

The DeskProto About Box will display the copyright notice and version number of your copy of DeskProto.

Do also note the Build date that is shown, which shows exactly which build of DeskProto you are working with. This is important in case support is needed. The format of the build data is “yyyy-mm-dd”. Several builds may exist having the same version number: the version number is increased only when the user interface has been changed (as then all translation files need to be changed too).

3.7.6 Context Sensitive Help

Use Context Sensitive Help to obtain help on some DeskProto user interface item. When you choose the Context Help button on the Toolbar, the mouse pointer will change to an arrow and question mark. Then click somewhere in the DeskProto window and the Help will be started showing the topic for the item you clicked.

Shortcut
Keys: SHIFT+F1

IV Concepts

4.1 Project

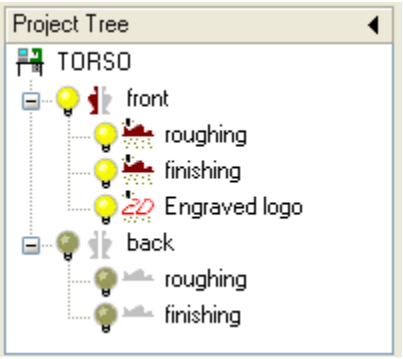
A project is comparable with a document in applications like MS Word. It is the main object you work with in DeskProto, it can be saved and opened in the File Menu, as a DeskProto ProJect file with the extension DPJ.

Structure
A project 'contains' a few other objects. One of the objects is the geometry. Each project can have either none or one geometry (which is in fact a link to a separate geometry-file). In case a project has no link to a geometry it is called a template project.

A second object is the part. A project can contain one or more parts (no limit). It is called part because it defines the shape of the part to be milled. You can for instance create a prototype by combining two separate parts, like left and right halve of a drilling machine. This part definition consists of a number of parameters, like scaling values (relative to the original geometry), rotation values etc. So the part parameters are in fact the geometry parameters.

The third object is called the operation. A part can contain one or more operations (no limit). It is called operation because it defines how a part should be made, how the material should be machined. This definition also consists of a number of parameters, like milling direction, precision values, feedrate etc. Many 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). So the operation parameters are in fact the milling parameters.
Three different types of operation are available: standard (3D) operations, and additional 2D Operations and bitmap operations.

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:



Besides geometry, parts and operations, a project contains a few other parameters: see the Project Parameters dialog for a description.

How to create a project
You can create a new project by choosing the New option of the File menu. When you create a new project the currently opened file will be closed. In case you start DeskProto a new project will be created automatically.

Default project
When a new project is created, the parameters are copied from the default project. The default project is saved somewhere in the computer (in the registry); every user has a separate set of defaults.

You can use the default project to enter parameters that you want to use as a standard. A default geometry file cannot be set, so these are only the checkboxes “Flip normals” and “Skip backfaces”. This will for instance be useful in case most of your STL files show backfaces..

To edit the parameters of the default project, go to the Options menu and select the option Default Project Parameters. This opens the Project Parameters dialog.

How to open a project
You can open a project by choosing the Open... option of the File menu. When you open a project the currently opened file will be closed.

The current ‘Look in’ folder in the Open File dialog, will be initiated conform the file location ‘Data’, which can be changed at the general tab page of the Preferences dialog.

How to save a project
You can save a project by choosing the Save option of the File menu. Using this option it will be saved under the same name you have saved it before. If you want to save it with another name, choose the option Save As... under the File men.

The current 'Save in' folder in the Save As dialog, will be initiated conform the file location 'Data', which can be changed at the general tab page of the Preferences dialog.

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 the geometry, only a link to the geometry file.

Two different types of DPJ files are used:
- a project file with only the settings (default)
- a project file with both settings and calculated toolpaths.
Both files have the same file extension .DPJ

The latter type can be used in case many Operations are combined in one NC program file, and the toolpath calculations take very long. Then when the settings are changed for only one operation, it is not needed to again calculate all operations to write the new NC program file.
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.

Note that in fact the toolpaths will be stored in a separate file: the **D**eskProto **T**ool**P**ath file (file extension **.DTP**), which will be much larger than the (small) DPJ file.
For a project file called test.dpj the toolpath file will have the name test.dpj. dtp
The difference between such Toolpaths file and an NC program file is that the first is in a machine-independent format.

4.2 Geometry

A project contains none or one geometry, or in other words it may link to one external geometry file.

Geometry type
DeskProto only recognizes geometry defined with triangles: the outer surface of the geometry described using a (large) number of triangles. This type of geometry is also called Polygon data. It is not the most efficient way of storing 3D data, still it has the large advantage that it always works: the level of definition is so low that no incompatibilities between systems exist.

This 3D geometry has to be distinguished from the optional 2D files that can be imported in a 2D Operation and a Bitmap Operation. 2D Contour data (a plotfile) is a flat vector-drawing and does by definition not contain any 3D Geometry. Bitmap data is a flat drawing defined using pixels, so does not contain 3D data either.

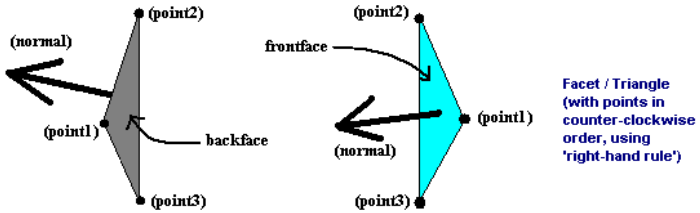
How to load a geometry
You can load the geometry file using the Load Geometry command in the File menu. Although DeskProto can only deal with one geometry per project, it is possible to load more than one geometry, using the same command: after you have loaded a geometry the Load Geometry option will be changed to Add Geometry. This is possible as DeskProto combines all the loaded geometry files to one new geometry.

How to save a geometry
Choosing the Save Geometry As option in the File, saves all the loaded geometries as one geometry file.

Filetypes
DeskProto supports three geometry filetypes: STL, DXF and VRML. The only type that is fully supported is STL, as that can only contain triangles. The other types may also contain other shapes (entities), which are not supported.

STL
STereoLithography File. Only contains triangles. An STL file may be a binary file or a ASCII file, the only difference being the storage method, the contents of both files (the series of triangles) is identical. DeskProto supports both types.
In an STL file for every triangle also one normal-vector is saved in order to define the inside and the outside of the part. This is in fact double

information, as also the order of the three points can be used to define inside and outside.



DXF
AutoCAD Drawing Interchange File. A DXF file may contain many different entities, both 2D and 3D.
Of all these entities only two are recognized by DeskProto: 3D-Faces and Polyface Meshes:
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 indices 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 2D DXF files, for 2D contour data in a 2D Operation.

VRML
Virtual Reality Modeling Language file. A VRML file may also contain many different entities and other objects. Only polygons and boxes are supported by DeskProto at this moment. All will be stored internally (and can be saved) as triangles.

Corrupt geometries
Unfortunately not all 3D CAD systems will export perfect geometry files, which may result in geometries that do not look right on the DeskProto screen. Note that with an up-to-date CAD program this is not likely to happen: most current CAD systems do have correct STL export functions.

A known problem with the STL files of some surface modeling CAD programs is incorrect normal-vectors. The normal-vector of a triangle defines

which side of the triangle is the outside, so incorrect normals result in a geometry with the **backfaces** on the outside, drawn in black. This can be solved by switching the option **Flip normals** on in the Project Parameters dialog.
You can also save your geometry with these flipped normals: choose the Save Geometry As option in the File menu.
STL-files do store the normal-vectors.
DXF files do not store the normals, when importing a DXF file DeskProto calculates the normal based on the three points of the triangle being stored in a counter-clockwise order.

When some of the normals are OK and some are incorrect, then only some of the triangles are drawn in black.. This can also be solved by DeskProto. You have to uncheck the option **Skip backfaces with calculations** in the Project Parameters dialog. This causes all the triangles to be treated as if they don't have backfaces but 2 frontfaces so they will be rendered from both sides, and both sides will be used for toolpath calculations as well.

Obviously files can be corrupt in many more ways. DeskProto is very tolerant here: it will just accept files containing cracks, holes, orphan surfaces and many more inconsistencies. Files that would be rejected by any other RP-system.

How to transform the geometry
For each part in your project you can set some transformations, like rotation, scale etc. These transformations can be set in the Part Parameters dialog at the Transform tab page.
Note:
The translation which may be needed to position the part on the milling machine can be set at the Translate tab page of the same dialog. These translation settings only effect the coordinates saved in the NC-program and the coordinates in the Geometry Information dialog under the Part section. All calculations are done using the untranslated coordinates, translation is applied when writing the NC file.
Note:
When you choose the Save Geometry As command in the File menu, you may select which of the transformations have to be applied for the exported STL file. You can do so in the Save Geometry Options dialog.

Viewing the geometry
You can view the geometry in different ways. As a wireframe, as a wireframe with hidden lines removed, as a rendered geometry or as points. To determine how to view the geometry you should check or uncheck the items in the Subjects dialog.

4.3 Part

A part is the thing that you will machine using one setup (a block being fixtured on the machine). Many models for instance can easiest be created by milling two separate Parts, for instance the left side and the right side of a hand drilling machine, to be glued together later. Also in case you machine one block of material from two 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 geometry), rotation values and the milling machine which will be used to produce the part. For a complete list of parameters see Part Parameters dialog.
How the part should be milled is described in its operations. A part can contain one or more operations.

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.

Current part
To view a particular part you need to make it **current**. There is always one part that’s current. No more (you cannot display two different geometries through one another), no less. The lightbulb icon in the project-tree shows which part is current: for only part the light burns (yellow) for the other parts (if any) the light is off (grey).
You can make a part current by clicking on the grey lightbulb icon for that particular part. Clicking on a yellow lightbulb 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. Another way to make a part current is in the dialog where you can choose which part you want to edit after selecting the option Part Parameters in the Parameters menu

Default part
When a new project is created, one part is added. The parameters of that part are copied from the ones set at the default part. The default part is saved somewhere in the computer (in the registry), this is done for every user who uses the computer.

You can use the default part to enter geometry parameters that you want to use as a standard. For instance the machine that you use, the translation method and the number of operations.

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.

4.4 Operation

An operation gives a description of how a part should be created, how the material should be machined. This Help page concerns DeskProto main operation type, which is the (3D) Operation. In addition an optional 2D Operation and Bitmap Operation are present.
The description of an operation consists of a number of parameters, like milling direction, precision values, feedrate etc. For a complete list of parameters see Operation Parameters dialog.

To edit an operation go to the Parameters menu and select the item Operation Parameters. This opens the Operation Parameters dialog. A much quicker way to open the Operation parameters dialog is double-clicking on the operation's name in the Tree.

Visible operations
To view the toolpaths (or segment, Z-grid etc) of a particular 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 lightbulb icon in the project-tree shows if an operation is visible: the light burns (yellow) means visible, the light is off (grey) means invisible.
You can make an operation visible by clicking on the grey lightbulb icon for that particular operation. You can also make an operation visible by right-clicking on an operation in the project-tree and then mark the option 'Visible' in the context-menu that will be shown. Another way to make an operation visible is when you check an operation in the operation-list of the Subjects dialog.

To actually see one of the items (Toolpath, Segment, Z-grid or other) of the operation you have made visible, you also should check the item you want to see in the Subjects dialog.

Default operation
When a new project is created, one part is added and one or more operations (depending on the number of operations in the default part) are added to that part. The parameters of the operations are copied from the ones set as default operations. The parameters of the first default operation are also used when you add a new operation to a part. The default operations have been stored somewhere in the computer (in the registry); this is done for every user who uses the computer.

You can use the default operation(s) to enter milling parameters that you want to use as a standard. For instance a specific tool, the accuracy and the strategy.

To edit the parameters of the default operations, go to the Options menu and select the option Default Operation Parameters. This will open the Operation Parameters dialog.

4.5 2D Data

2D versus 3D
DeskProto is aimed at working with a 3D Geometry, and most milling strategies and other parameters are only suited for 3D applications. Still many users also will need simple 2D machining, either for simple parts or for some special detail in a 3D project. Thus DeskProto also offers options for simple 2D machining. These are clearly additional functions: DeskProto does not aim to be a true 2D CAM system.

A DeskProto project contains a link to one 3D geometry file, which is used for all parts and all operations in the project. In contrast a 2D file can be used only for one operation. It contains 2D Contour data, which is a flat line-drawing, and the cutter will just follow these 2D lines on a certain Z-level. This use should be seen as an (old) pen plotter: the lines in the plotfile are plotted on **pen-down** Z-level, with in-between positioning moves on **pen-up** Z-level. Note that the center of the cutter follows the line: for 2D DeskProto does not apply cutter compensation (compensate for the radius of the cutter). So a DeskProto project contains (links to) none, one or more 2D files.

To support 2D machining a special 2D Operation has been added. A Part may contain both 2D Operations and 3D Operations: each 2D operation having its own 2D file, and each 3D Operation using the projects geometry.
So in order to use 2D machining you will have to add a 2D Operation first.

How to load a 2D File
After having created a 2D Operation, you can load a 2D file in the 2D Operation Parameters using the [Browse…] button for a standard Windows File-Open dialog.

2D Filetypes
For 2D DeskProto supports two filetypes: **DXF** as this is the most widely used format for engineering, **Postscript** as that is the most widely used format for Graphics design. Just as for 3D DXF files, for both 2D filetypes only a subset is supported.

DXF
AutoCAD Drawing Interchange File. Can contain many different entities, both 2D and 3D, DeskProto supports a small subset only. Note that the supported entities (listed below) may also contain Z-information which in fact makes them 3D. DeskProto's 2D file load will ignore any Z-coordinates in the DXF file.

Supported DXF entities for 2D in DeskProto:
Point
A point in the DXF file will result in a drilling operation on that location.
Line
A line contains a begin- and end-point, and will result in a tool movement from begin to end (linear interpolation).
Polyline and **LW polyline**
A polyline is in fact a series of lines connected to each other. This is more efficient (endpoint line 1 = startpoint line 2) and gives more control over the toolpath sequence. A polyline may contain arc segments as well.
Arc
An arc entity contains the centerpoint coordinates, the radius of the arc, and angles for start and stop. Currently DeskProto will convert the arc to many small line segments, as the toolpath part (postprocessor) does not yet support arcs.
Circle and **Ellips**
Same as the Arc: stored in DXF as Centerpoint(s) and radius, converted by DeskProto to polylines.

In practice many DXF-files will also contain other entities, which will be skipped by DeskProto. A Line, Arc or Circle can for instance also be stored in the DXF as a spline curve: DeskProto will fail to read that entity. This is for instance critical for text as most fonts are defined with splines. Try to find an export option that converts the splines to polylines. Also text that is stored as ASCII character with a font definition cannot be read by DeskProto.

Note that DeskProto also supports 3D DXF files, for 3D Geometry.

Postscript
In fact the correct name is Encapsulated Postscript, file extension .EPS or . AI (short for Adobe Illustrator). For Postscript files as well DeskProto supports a subset only.

Supported Postscript entities for 2D 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
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.

Postscript also may contain many other entities, like colors and bitmaps, which will be skipped by DeskProto.

How to transform the 2D File data

The 2D contour data from the 2D file will be interpreted by DeskProto in machine coordinates, so relative to the workpiece zero point as used on the machine. The milling machine will indeed be used as a plotter and let the cutter move to the coordinate values exactly as specified in the 2D file.

The machine coordinates are shown on the Part tab of the Geometry Information dialog. This means that any transformations (rotation, scale,…) and translations that you have set in the 3D Part Parameters do **NOT** apply on the 2D contour data. The logic is that some transformations, like 3D rotation and inverse milling, do not make sense for 2D plotdata.

You can transform the 2D contour data on the XY transform Tab of the 2D Operation parameters. This dialog offers options to scale, translate and rotate, all in 2D. Using the apply button you can easily position the 2D contour data as needed, using a step-by-step approach. As an alternative an Align dialog is available for your convenience.

Note:

This different treatment of a 3D geometry file and a 2D plotfile might give a problem when you want to combine both file types for a part. This problem can easily be solved, by not applying any Transformation to the 3D part, and by setting the 3D Translation to None for all axes. Also no 2D transformations may be applied. Then the 3D and the 2D file (in case written in the same coordinate system, will nicely match.

A different type of Transform is offered on the Z-settings Tab page, which allows you to transform the Z-values by projecting the 2D contour data on the 3D geometry.

Viewing the 2D File data

You can see the 2D contour data on your graphics screen. In case you do not see the plotfile, check:

- does the 2D operation contain a (link to a) 2D file.
- is the 2D operation visible
- is the entry “2D contours” checked in the Subjects dialog.

4.6 2D Operation

An operation gives a description of how the material should be machined. This Help page concerns one of DeskProto's additional operation types: the **2D Operation**. The main operation type is the (3D) Operation.

You have to be aware that DeskProto only supports very basic 2D functionality: for more advanced 2D options a specialized 2D CAM software is needed. Basically in 2D mode DeskProto acts like a plotter. The cutter will exactly follow the lines as defined in the 2D file, at a certain machining depth (pen-down level). For positioning moves in-between the cutter rises to Z-free height (pen-up level).
In 2D mode DeskProto does not compensate for the cutter radius: so when you machine a square of 100 mm using a cutter of 10 mm diameter (R 5), the result will be a square of 100 – (2 x 5) – 90 mm large. Also options like pocketing and start/stop sequences are not available.

The description of a 2D operation consists of a number of parameters, like milling direction, precision values, feedrate etc. For a complete list of parameters see 2D Operation Parameters dialog.

To edit a 2D Operation go to the Parameters menu and select the item Operation Parameters. This opens a dialog to select which Operation to edit. After selecting a 2D Operation, DeskProto will show the 2D Operation Parameters dialog. Of course a 2D Operation must have been added already.
A much quicker way to open the 2D Operation parameters dialog is double-clicking on that operation's name in the Tree.

Default 2D operation
When a new 2D Operation is created, the parameters of the new operations are copied from the ones set as default 2D operation. This default 2D Operation has been stored somewhere in the computer (in the registry), this is done for every user who uses the computer.
DeskProto also contains a default project and a default part. The default part may also contain a 2D Operation: in that case the default 2D Operation will also be used when creating a new Project and/or a new Part.
You can use the default 2D 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 2D operation, go to the Options menu and select the option Default 2D Operation Parameters. This will open the 2D Operation Parameters dialog.

4.7 Bitmap Data

Bitmap versus 2D and 3D

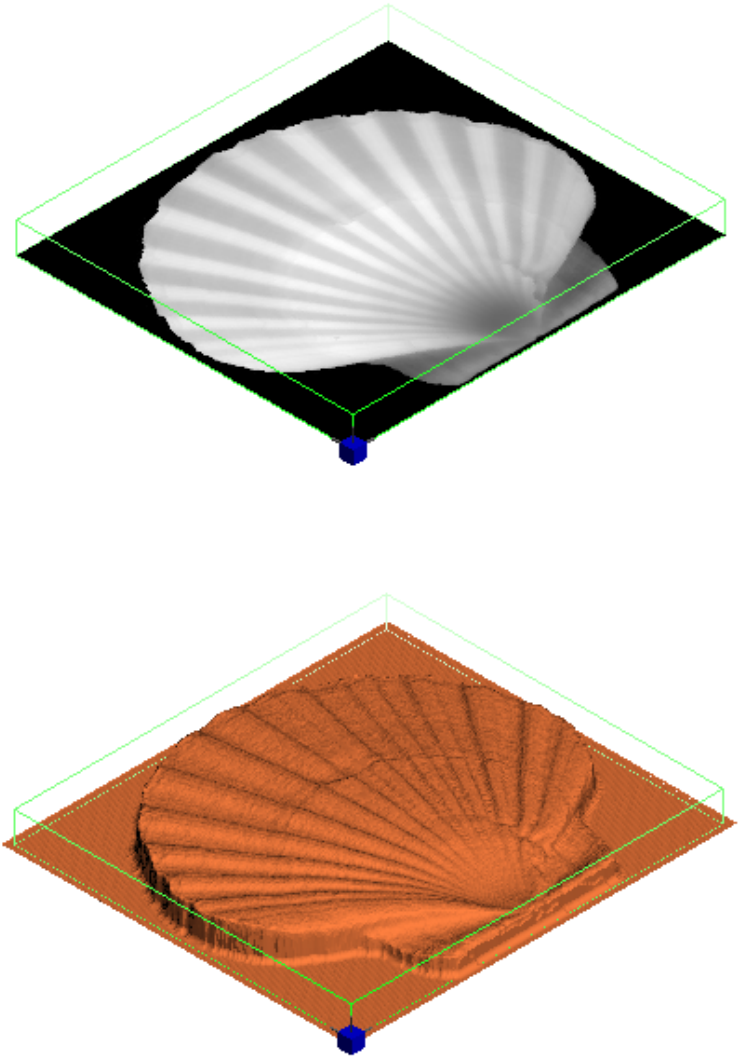
A bitmap file contains a 2D picture by storing the color value of each pixel on the screen. So a line is stored as a series of black pixels on a white background. This in contrast to a vector file, where the line is stored using the coordinates of both the start point and the end point of the line. As many different colors can be used, in a bitmap complex pictures are possible. Digital photos for instance are stored in bitmap files.

A bitmap file is a 2D file: a pixel only has an X-coordinate and a Y-coordinate, plus of course a color. DeskProto converts this 2D file to 3D by translating the color 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 colors range between black and white, defining the extreme Z-values to be used. Color pictures are automatically converted to Black & White (conversion formula: Grey-value = 0.30 * red + 0.59 * green + 0.11 * blue). The extreme Z-values (for pure black and pure white) can be set in the Bitmap Settings dialog. Internally DeskProto converts the bitmap into a Z-grid, that can be used for toolpath calculations.

This is an exception to the basic DeskProto method of working with a 3D Geometry, when the geometry is created using a CAD program or a 3D scanner and then imported by DeskProto. Such 3D Geometry cannot be changed except some basic transformations. Bitmap geometries in contrast are created in DeskProto. The toolpath options for bitmaps are roughly the same as for a standard 3D Operation.
The approach also differs from a 2D Operation as there data and toolpaths both are in 2D.

A DeskProto project contains (a link to) one 3D geometry file, which is used for all parts and all operations in the project. In contrast the link to a bitmap file is set at operation level, so several bitmaps can be used in the same project.

To support bitmap machining a special Bitmap Operation has been added. A Part may contain 3D Operations, 2D Operations and/or Bitmap Operations: each 3D Operation using the projects geometry, each 2D Operation using its own 2D file and each Bitmap Operation using its own Bitmap file.
So in order to use Bitmap machining you will have to add a Bitmap Operation first.



The two pictures above show the original bitmap image of a shell left, and right a simulation of the resulting 3D Relief. Min Z has been assigned to black, and Max 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.

How to load a Bitmap File
After having created a Bitmap Operation, you can load a Bitmap file in the Bitmap Operation Parameters using the [Browse…] button for a standard Windows File-Open dialog.

Bitmap Filetypes
DeskProto supports the three most popular filetypes for bitmap files:
BMP File format defines by Microsoft for **B**it**M**a**P** files. Not very efficient (large files), widely accepted.
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.
GIF the **G**raphic **I**nterchange **F**ormat is an efficient format without data loss, though only max 256 colors are possible.

How to transform the Bitmap File data
The Bitmap geometry will be created from the Bitmap file in machine coordinates, so relative to the workpiece zero point as used on the machine. So Part transformations and Translations do **not** influence the Bitmap geometry. The default XY position of the bitmap will be with the left bottom corner on point (0,0), Z-positions as specified in the Bitmap settings. In these same Bitmap settings the XY size is defined, and a Translation for X and Y may be specified in case needed. Using the apply button and looking on the graphics screen you can easily position the bitmap geometry as needed, using a step-by-step approach.
As an alternative an Align dialog is available for your convenience.

A different type of Transform is offered on the Bitmap Z Tab page, which allows you to transform the Z-values by projecting the 2D contour data on the 3D geometry.

Viewing the Bitmap File data
You can see the Bitmap image on your graphics screen. Note that a color image is shown in black & white. In case you do not see any image, please check:
- does the bitmap operation contain a (link to a) bitmap file.
- is the bitmap operation visible
- is the entry “Bitmaps” checked in the Subjects dialog.

4.8 Bitmap Operation

An operation gives a description of how the material should be machined. This Help page concerns DeskProto's additional operation type called the **Bitmap Operation**. The main operation type is the (3D) Operation, and a second additional one is the 2D Operation. The Bitmap operation does not use the Part geometry, but uses a Bitmap geometry instead.

The description of a Bitmap operation consists of a number of parameters, like milling direction, precision values, feedrate etc. For a complete list of parameters see Bitmap Operation Parameters dialog. You can find this via Operation Parameters in the Parameters menu. Of course a Bitmap Operation must have been added first.
A much quicker way to open the Bitmap Operation parameters dialog is double-clicking on that operation's name in the Tree.

As for a bitmap operation also parameters are needed for the conversion from bitmap image to bitmap geometry, an extra dialog has been added called the Bitmap Settings dialog.

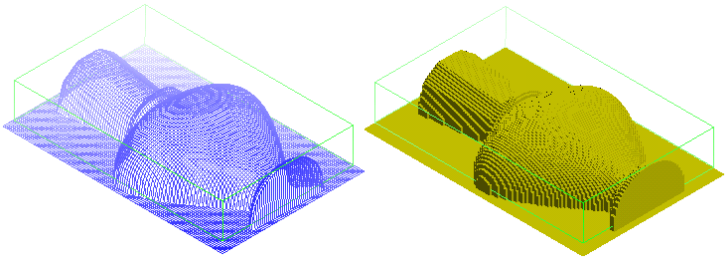
Default Bitmap operation
When a new Bitmap Operation is created, the parameters of the new operations are copied from the ones set as default Bitmap operation. This default Present Operation has been stored somewhere in the computer (in the registry), this is done for every user who uses the computer.
DeskProto also contains a default project and a default part. The default part may also contain a Bitmap Operation: in that case the default Bitmap Operation will also be used when creating a new Project and/or a new Part.
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 machining depth.

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 Default Bitmap Operation Parameters dialog. One of the options in this dialog leads to the Default Bitmap Settings dialog.

4.9 Z-Grid

The Z-grid is an intermediate result, in-between the geometry and a toolpath. The Z-grid is a sort of height-map of the geometry: a 3D bar graph, with a Z-value for each XY position in a grid as specified by the toolpath distance and the stepsize along the toolpath. Normally you should not be bothered by this representation of the geometry, however it might be useful to see what is really happening. For instance in case a hole in the geometry was missing in the toolpath, you could check the Z-grid. In case present there, the cause will be that the cutter is too large to fit in the hole. In case not present in the Z-grid, some error is present is the geometry file. Besides that it can also give an idea of how the final result will look: a rough simulation of the part to be created. Especially when you view it as a rendered Z-grid.

Here's an image with a Z-grid and the rendered one (do note that a rendered Z-grid for an accurate toolpath takes some time to draw on your screen).



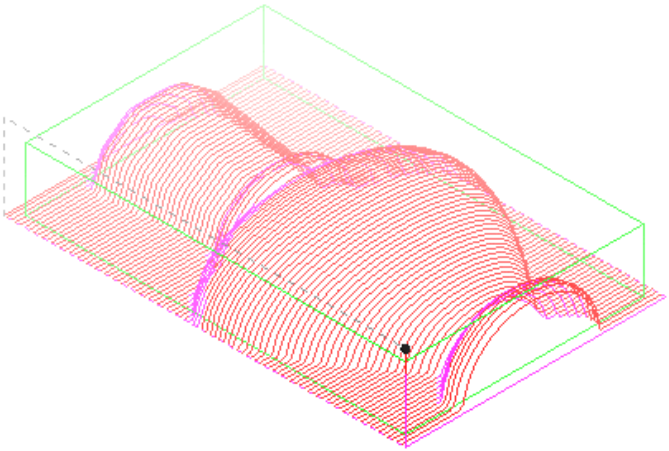
The pictures show a line-drawing and a rendered Z-grid (to be displayed using the "Subjects in View" dialog) The grid-structure is clearly visible. It is in fact a simulation of the part that will be actually machined. Do note that all undercuts present in this bottle (there is one in it's bottom) will become solid. Especially for inverse milling this rendered Z-grid is useful as a simulation to show what will be created. Do also note that a very strong staircase effect is visible. In the actual model the stairs will be partially smoothed away due to the size of the cutter (as it cannot create sharp inner corners).

How to calculate it
You can let the Z-grids be calculated by selecting the option Calculate Z-grids under the Create - Extra menu. Another way of doing it is checking the Z-Grids checkbox or the Rendered Z-Grids checkbox in the Subjects dialog.

4.10 Toolpath

The Toolpath is the path that the cutter will follow to create a model on the milling machine. More specific, it is the series of positions for the **tip** of the cutting tool (in XY the center of the tool, in Z 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 Subjects dialog an option is present to display an extra orientator at the workpiece zero point.

The toolpath is a very important representation for a visual check before starting the milling machine: any possible errors should be found before milling. Here's an image of a toolpath.



Most lines are drawn as solid lines (default in red), indicating a normal speed (Feedrate). Some are dashed in grey, indicating that they are **Rapid** movements (moving as fast as possible). The toolpath depends on the settings made in the Operation Parameters dialog.

After calculating a toolpath using the command **Calculate toolpaths** the toolpath will automatically be switched on in the scene (made visible, done in the Subjects dialog). To send this toolpath to the machine it should be post-processed to an NC-program, which is done using the command **Write NC-program file**. How it will be post-processed is determined by the postprocessor which is linked to the milling machine you use.

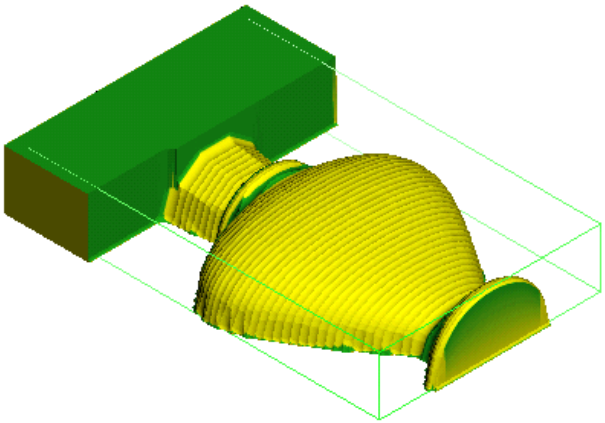
How to calculate it
You can let the toolpaths be calculated by selecting the option Calculate Toolpaths under the Create - Extra menu.

As a shortcut this convenient button is present.
Another way of doing it is checking the Toolpaths checkbox in the Subjects dialog.

4.11 Simulation

The NC program file is in fact the final result of DeskProto: sending it to the machine will produce the desired prototype. It may be useful though to preview this resulting prototype on screen, in a machining Simulation.

The Simulation is a drawing on screen that shows you what the resulting part will be. This can be used to check things like the resulting surface smoothness, error movements (if any) that damage the part, rest material where the cutter cannot reach, etc. DeskProto will calculate a simulation in 3D, so you can rotate, pan and zoom it just like any other subject on screen. Currently the DeskProto simulation can do one operation at a time, and only for 3 axis machining.
You can calculate a simulation by using the Calculate Simulation command in the Create menu. You can also switch the simulation on and off in the Subjects in view Dialog to have a simulation calculated and shown.



The example drawing above of a Bottle Toolpath clearly shows the rest material in-between the ballnose toolpaths (such ridge is called cusp). In the example the Simulation calculation is interrupted halfway by pressing the button Show Results Now in the Simulation calculation's progress bar. You can exactly see how far the cutter has come.

The DeskProto simulation allows you to compare the simulation with the original geometry, and indicate any differences with a color. Rest material in green, too much material machined in red. In the Operation parameters the user can select whether or not to use these colors, and also the tolerance to be used.

In the example drawing above you can for instance see that the cutter is too thick for the small inner radius at the neck of the bottle (just below the cap).

At that location rest material is left over, indicated in green. Note that the red color is only visible in case the geometry is switched off in the Subjects dialog.

How to calculate it
You can let the Simulations be calculated by selecting the option Calculate Simulation under the Create menu. Another way of doing it is checking the Simulations checkbox in the Subjects dialog.

4.12 NC-program

NC stands for Numerical Control (we will also use CNC, which stands for Computerized Numerical Control). An NC-program is a file that contains an NC program: a series of instructions for a CNC milling machine to execute. It can be sent to a machine to make it execute the NC program. Each file contains commands for a specific machine (or machine-type). So an NC-program is machine-dependent: its format will be different per machine. In general it is a file which can control an NC-machine.
Almost any NC program file is in plain ASCII: you can read and edit it using a plain text editor like Notepad.

DeskProto will try to combine the toolpaths of all Operations of one Part into one combine NC program file.
Or, if Chaining has been applied, even toolpaths of Operations in several Parts can be combined.

How to create it
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.

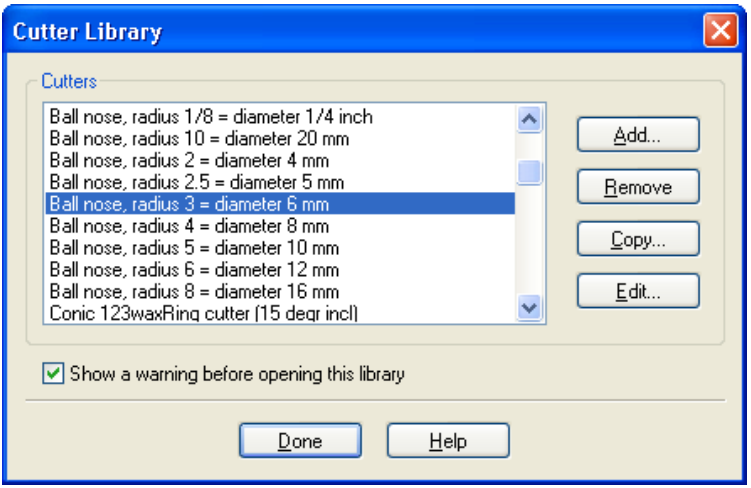
NC Files list
DeskProto can show you a list of all NC files that have been written for the current project. This is shown in the NC Files window, below the Project Tree. If this window is not visible you can check it in the View menu.

Selecting machine and postprocessor
The format of the NC-program you produce will be determined by the postprocessor which is linked to the machine you use. You can select this machine at the Part Parameters dialog.

Sending NC-program to machine
For some machines it's possible to send an NC-program to the machine from within DeskProto. How to do this see the option Send NC-program To Machine from the Create menu.

4.13 Libraries

There are three libraries. One for machines, one for postprocessors and one for cutters.
Each library is displayed in a dialog like this:



A library is just a collection of definitions. You can either add, copy, remove or edit an item. The definitions are saved to disk in the file location Drivers, which can be set at the General tab page of the Preferences dialog box. They are saved to disk when you click the OK-button of the library-dialog.

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 do want to change a filename of a cutter, postprocessor or machine you can use Windows Explorer to do so. Preferably you do this when DeskProto is not running (so close DeskProto first) because otherwise the old filename will be saved again when closing the library. The file with the new name will be automatically loaded when you restart DeskProto.

See Library Of Machines, Library Of PostProcessors and Library Of Cutters.

All these definitions are stored in files (machines in .mch files, postprocessors in .ppr files and cutters in .ctr files), to be stored in the Drivers directory of DeskProto. All these files are plain ASCII text files, that can also be edited using a plain text editor.
For editing no help information is present, so only do so if you are sure what you are doing.
From Version 4.1 it is possible to use comments in these files: any line that

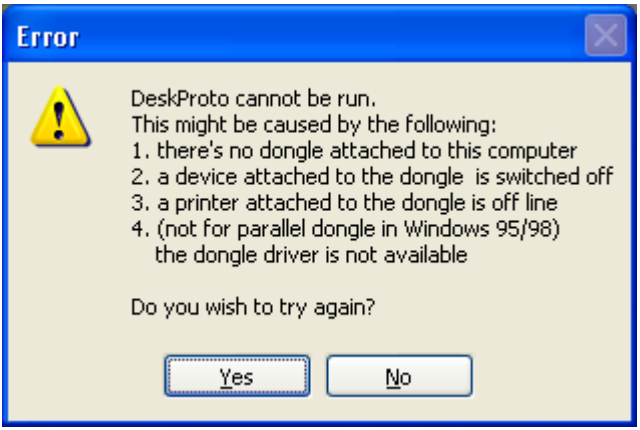
starts with a semicolon (“;”) is considered comment and thus ignored by DeskProto.

V Troubleshooting

5.1 Dongle problems

DeskProto Full is protected by a Dongle: a small piece of hardware that needs to be plugged into the PC as otherwise DeskProto will refuse to start. This does not apply to the Trial version, nor to DeskProto Lite. Two types of dongle do exist: for the parallel port (printer port) and for the USB port. The first type is in fact obsolete, and can be exchanged for a USB dongle when ordering an update to a next version of DeskProto.

Warning:
Be careful not to loose your dongle: lost or stolen dongles will NOT be replaced.



When no DeskProto Dongle can be found the above error message is displayed. Below you will find some suggestions in case of Dongle problems. The list has three sections: General, Parallel and USB.

Note:
For checking the dongle you can use the dongle diagnostics program DDLOOK.EXE which can be found on the DeskProto CD (directory \Dongle\).

General dongle problems:

1. You are using a dongle which belongs to another program.

Solution:
Make sure you're using the dongle that has been delivered with DeskProto (should be labeled "DeskProto").

2. The dongle has been removed after DeskProto has been started.
Solution:
Keep the dongle attached during the whole DeskProto session.

3. The dongle has been damaged.
Solution:
In case you cannot solve the problem, write down the error-message that occurred and contact your dealer or Delft Spline Systems for support. We will send you a new dongle on receipt of a damaged one.

USB dongle problems:

4. The dongle drivers are not present or have not correctly been installed. Note: the green LED in the USB dongle has to burn, if not the drivers have not been installed correctly.
Solution:
See the next paragraph, on USB Dongle Drivers

5. The Windows Plug-and-Play wizard has installed incorrect drivers. Note: to prevent this it is needed to FIRST install DeskProto, and THEN plug in the USB dongle.
Solution:
See the next paragraph, on USB Dongle Drivers

Parallel dongle problems:

Since many years already DeskProto is shipped with a USB dongle, only users that have bought their DeskProto long time ago may still have a parallel dongle.
They need to pay attention: by default the DeskProto 5.0 Setup installs dongle drivers for the USB dongle only. On an old PC that is no problem (the parallel driver already is present), on a new PC it is. See problem 6.

6. The dongle driver is not present: DDlook returns error 125 "Cannot communicate with device driver". This may happen as by default the DeskProto Setup only installs the USB dongle drivers. In order to also install the parallel dongle driver:
- insert the DeskProto CD in the CD drive
- in the Run command (Start button) enter "E:\dongle\setupdrv.exe /par" (E: may be a different character on your PC).

7. The dongle is not (or not well enough) attached.
Solution:
Attach the dongle to the LPT-port (printer-port)and screw it tightly.

8. A printer or other device that is connected to the same LPT-port is switched off. With some printers this might be a problem.
Solution:
Make sure that the device attached to the same LPT-port is switched on.

9. A printer device that is connected to the same LPT-port is ‘off line’.
Solution:
Make sure that any printer device attached to the same LPT-port is ‘on line’.

10. A laptop computer is used that puts insufficient voltage on the LPT port to let the dongle function correctly.
Solution:
Attached a printer to the same LPT-port (so on the dongle) and switch it on.

11. The dongle drivers are not present or have not correctly been installed.
Solution:
Let DeskProto be reinstalled by someone who has ‘administrator’ rights.

5.2 USB Dongle Drivers

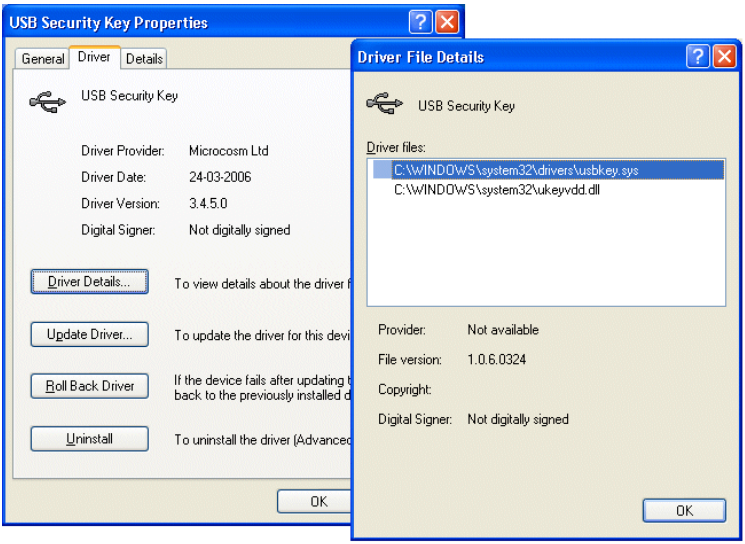
In case you are troubleshooting in order to solve a problem with your dongle, please first read the section on Dongle Problems. You can also first visit the DeskProto FAQ (list of Frequently Asked Questions) on the website www.deskproto.com This Faq includes a few dongle issues in section " Problems when using DeskProto".

The following text explains how to change the drivers for a USB dongle in case needed. This may be needed in case you have plugged the USB dongle into the PC before running the DeskProto Setup. When plugging in an unknown USB device, Windows will start the **New Hardware Found Wizard** and try to install drivers for the device. However if these could not yet be found as the correct drivers are provided by the DeskProto Setup, Windows might have installed incorrect drivers.

You can easily see whether or not the correct drivers are used: in case the dongle connects to the correct driver, **the green led in the dongle will light**. So if you do not see a green led (PC has to be on) in the inserted dongle, you have a problem. In almost all cases this will be a driver problem.
Parallel dongles do not have a LED. The driver is also less complicated, most problems with parallel dongles are hardware related (insufficient power).

Here is what you can do to load new drivers for your USB dongle, in case the LED does not light. The instructions below are for Windows XP. Unfortunately the location of the Device Manager is different per Windows version, so in case you are not using Win XP some searching may be needed. In all versions you can find it in the System Properties of the Control Panel.

- Open the **Control Panel** (Start menu, Settings).
- Start **System Properties** by double clicking System
- Go to Tab Hardware and press button **Device Manager**
- Press the + in front of **Software Security Token** and
- Double-click **USB Security Key**
- Go to Tab Driver and press button Driver Details.



The Driver Tab and Driver File Details dialog show which driver file has been installed. If it does not match the above illustrations (Provider needs to be Microcosm), then you have an incorrect driver installed. Strange is that the Driver File Details dialog not always shows the Provider.
In case the wrong driver has been installed, press OK to close the Driver Details dialog, and next press the button **Update Driver...**

Now the **Hardware Update Wizard** pops up, which closely resembles the New Hardware found Wizard as explained in the Installation chapter of the DeskProto Tutorial.
- Insert the DeskProto CD
- Close the DeskProto Install menu that pops up
- Choose **No, not this time** in the Wizard for Connect to Windows Update and press Next.
- Choose **Install from a list or a specific location** and press Next.
- Let the wizard **Search removable Media** (CD ROM) and press Next.

Now the wizard will install the drivers from your CD, as for this specific purpose the drivers are present on the root of the DeskProto CD.
After finishing the wizard, restart your PC: now the green LED in your dongle should be on.

When the above does not help, the last option is to completely uninstall the existing drivers and install new.

The DeskProto dongle drivers normally are provided by the program SetupDrv.exe , to be found on the DeskProto CD. This driver setup program is called automatically by the DeskProto Setup program, so normally users do not need to know about it. After completing a DeskProto Setup and then inserting the dongle, the New Hardware found wizard can be completed using “Install the software automatically”, as explained in the Tutorial.

This same program SetupDrv can also be used to un-install these drivers in case needed. You can call SetupDrv with a command line parameter using the Run button in the Windows Start menu.
Here is what you need to do:

- First **remove all dongles** from the PC.
- Run **Setupdrv /ufull** (this will remove the current drivers).
- **Reboot** the PC.

Next you can again install (correct) drivers:

- Run SetupDrv
- Insert the Dongle
- Follow the New Hardware found Wizard (see the DeskProto Tutorial for Help).

5.3 Runtime error

A runtime error can occur in the following situations.

1. The DeskProto kernel dll ‘DPKERNEL.DLL’ is missing.
Solution:
In case the DPKERNEL.DLL is not present anymore, reinstall DeskProto.

2. The DeskProto kernel dll ‘DPKERNEL.DLL’ has been moved to another location.
Solution:
Make sure the DPKERNEL.DLL is placed in the same directory as DP.EXE

3. When this still does not help, do the following.
1. Go to the ‘Start’ menu
2. Select ‘Run’
3. Type in the following:
regsvr32 “C:\Program Files\DeskProto 5.0\DpKernel.dll”
(in case the dll is located in another directory, replace the directory specification in this example command by the correct specification).
4. Press the OK-button

4. Unknown situations.
Solution:
In case it still does not work, uninstall DeskProto and reinstall it again.

5.4 Trial period problems

After the trial period of 30 days has expired, DeskProto will run with limited functionality. NC-program files can no longer be saved. Note that you still can save project files (DPJ files), so you can use the free trial license to prepare a project and then use the paid license to calculate the NC file. You also still can use the trial program as free STL file viewer, and also as format converter (DXF to STL and more).

In some cases though DeskProto runs with limited functionality while your 30 days trial period has not been passed. There are two possible causes:

1. The system date has been set to a future date after installation of the DeskProto Trial version (later than the end end of the trial period). DeskProto now thinks your 30 days already have past. The trial period can not (!) be reset again.
Warning:
Be sure that the date is set correctly both before you install DeskProto Trial version and before you run the DeskProto Trial version.

2. The system date or time has been set backwards after installation of the DeskProto Trial version. DeskProto will detect this, making it impossible to increase your trial period by doing that.
Solution:
In case the system date or time has been set backwards in time, reset the correct date before you run the DeskProto Trial version.

As the 30 days trial protection has been implemented using a third party Software Development Kit, we have no tools to reset or prolong the 30 days.

VI How to ...

6.1 Deal with 2D Files

6.1.1 Information on Bitmap files

Bitmap versus 2D and 3D

A bitmap file contains a 2D picture by storing the color value of each pixel on the screen. So a line is stored as a series of black pixels on a white background. This in contrast to a vector file, where the line is stored using the coordinates of both the start point and the end point of the line. As many different colors can be used, in a bitmap complex pictures are possible. Digital photos for instance are stored in bitmap files.

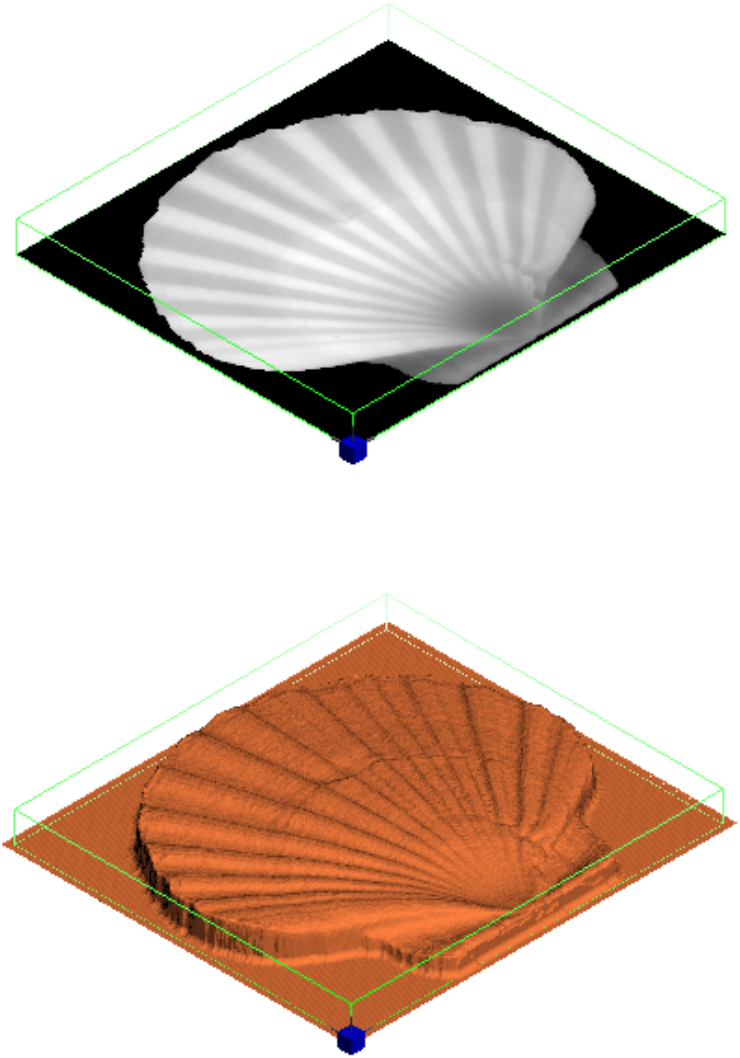
A bitmap file is a 2D file: a pixel only has an X-coordinate and a Y-coordinate, plus of course a color. DeskProto converts this 2D file to 3D by translating the color to a Z-value, resulting in XYZ information for each pixel. The result is a 3D Relief. This is called **Gray scale to Z-height conversion**, or **Bitmap to Relief conversion** The colors range between black and white, defining the extreme Z-values to be used. Color pictures are automatically converted to Black & White (conversion formula: Grey-value = 0.30 * red + 0.59 * green + 0.11 * blue). The extreme Z-values (for pure black and pure white) can be set in the Bitmap Settings dialog. Internally DeskProto converts the bitmap into a Z-grid, that can be used for toolpath calculations.

This is an exception to the basic DeskProto method of working with a 3D Geometry, when the geometry is created using a CAD program or a 3D scanner and then imported by DeskProto. Such 3D Geometry cannot be changed except some basic transformations. Bitmap geometries in contrast are created in DeskProto. The toolpath options for bitmaps are roughly the same as for a standard 3D Operation.
The approach also differs from a 2D Operation as there data and toolpaths both are in 2D.

A DeskProto project contains (a link to) one 3D geometry file, which is used for all parts and all operations in the project. In contrast the link to a bitmap file is set at operation level, so several bitmaps can be used in the same project.

To support bitmap machining a special Bitmap Operation has been added. A Part may contain 3D Operations, 2D Operations and/or Bitmap Operations: each 3D Operation using the projects geometry, each 2D Operation using its

own 2D file and each Bitmap Operation using its own Bitmap file.
So in order to use Bitmap machining you will have to add a Bitmap Operation first.



The two pictures above show the original bitmap image of a shell left, and right a simulation of the resulting 3D Relief. Min Z has been assigned to

black, and Max 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.

How to load a Bitmap File
After having created a Bitmap Operation, you can load a Bitmap file in the Bitmap Operation Parameters using the [Browse…] button for a standard Windows File-Open dialog.

Bitmap Filetypes
DeskProto supports the three most popular filetypes for bitmap files:

BMP File format defines by Microsoft for **B**it**M**a**P** files. Not very efficient (large files), widely accepted.

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.

GIF the **G**raphic **I**nterchange **F**ormat is an efficient format without data loss, though only max 256 colors are possible.

How to transform the Bitmap File data
The Bitmap geometry will be created from the Bitmap file in machine coordinates, so relative to the workpiece zero point as used on the machine. So Part transformations and Translations do **not** influence the Bitmap geometry. The default XY position of the bitmap will be with the left bottom corner on point (0,0), Z-positions as specified in the Bitmap settings. In these same Bitmap settings the XY size is defined, and a Translation for X and Y may be specified in case needed. Using the apply button and looking on the graphics screen you can easily position the bitmap geometry as needed, using a step-by-step approach.
As an alternative an Align dialog is available for your convenience.

A different type of Transform is offered on the Bitmap Z Tab page, which allows you to transform the Z-values by projecting the 2D contour data on the 3D geometry.

Viewing the Bitmap File data
You can see the Bitmap image on your graphics screen. Note that a color image is shown in black & white. In case you do not see any image, please check:
- does the bitmap operation contain a (link to a) bitmap file.
- is the bitmap operation visible
- is the entry “Bitmaps” checked in the Subjects dialog.

6.2 Deal with Milling parameters

6.2.1 Part principles

A part is the thing that you will machine using one setup (a block being fixtured on the machine). Many models for instance can easiest be created by milling two separate Parts, for instance the left side and the right side of a hand drilling machine, to be glued together later. Also in case you machine one block of material from two 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 geometry), rotation values and the milling machine which will be used to produce the part. For a complete list of parameters see Part Parameters dialog.
How the part should be milled is described in its operations. A part can contain one or more operations.

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.

Current part
To view a particular part you need to make it **current**. There is always one part that's current. No more (you cannot display two different geometries through one another), no less. The lightbulb icon in the project-tree shows which part is current: for only part the light burns (yellow) for the other parts (if any) the light is off (grey).
You can make a part current by clicking on the grey lightbulb icon for that particular part. Clicking on a yellow lightbulb 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. Another way to make a part current is in the dialog where you can choose which part you want to edit after selecting the option Part Parameters in the Parameters menu

Default part
When a new project is created, one part is added. The parameters of that part are copied from the ones set at the default part. The default part is saved somewhere in the computer (in the registry), this is done for every user who uses the computer.

You can use the default part to enter geometry parameters that you want to

use as a standard. For instance the machine that you use, the translation method and the number of operations.

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.

6.2.2 Operation principles

An operation gives a description of how a part should be created, how the material should be machined. This Help page concerns DeskProto main operation type, which is the (3D) Operation. In addition an optional 2D Operation and Bitmap Operation are present.
The description of an operation consists of a number of parameters, like milling direction, precision values, feedrate etc. For a complete list of parameters see Operation Parameters dialog.

To edit an operation go to the Parameters menu and select the item Operation Parameters. This opens the Operation Parameters dialog. A much quicker way to open the Operation parameters dialog is double-clicking on the operation's name in the Tree.

Visible operations
To view the toolpaths (or segment, Z-grid etc) of a particular 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 lightbulb icon in the project-tree shows if an operation is visible: the light burns (yellow) means visible, the light is off (grey) means invisible.
You can make an operation visible by clicking on the grey lightbulb icon for that particular operation. You can also make an operation visible by right-clicking on an operation in the project-tree and then mark the option 'Visible' in the context-menu that will be shown. Another way to make an operation visible is when you check an operation in the operation-list of the Subjects dialog.

To actually see one of the items (Toolpath, Segment, Z-grid or other) of the operation you have made visible, you also should check the item you want to see in the Subjects dialog.

Default operation
When a new project is created, one part is added and one or more operations (depending on the number of operations in the default part) are added to that part. The parameters of the operations are copied from the ones set as default

operations. The parameters of the first default operation are also used when you add a new operation to a part. The default operations have been stored somewhere in the computer (in the registry); this is done for every user who uses the computer.

You can use the default operation(s) to enter milling parameters that you want to use as a standard. For instance a specific tool, the accuracy and the strategy.

To edit the parameters of the default operations, go to the Options menu and select the option Default Operation Parameters. This will open the Operation Parameters dialog.

6.3 Deal with viewing

6.3.1 Which subjects to view

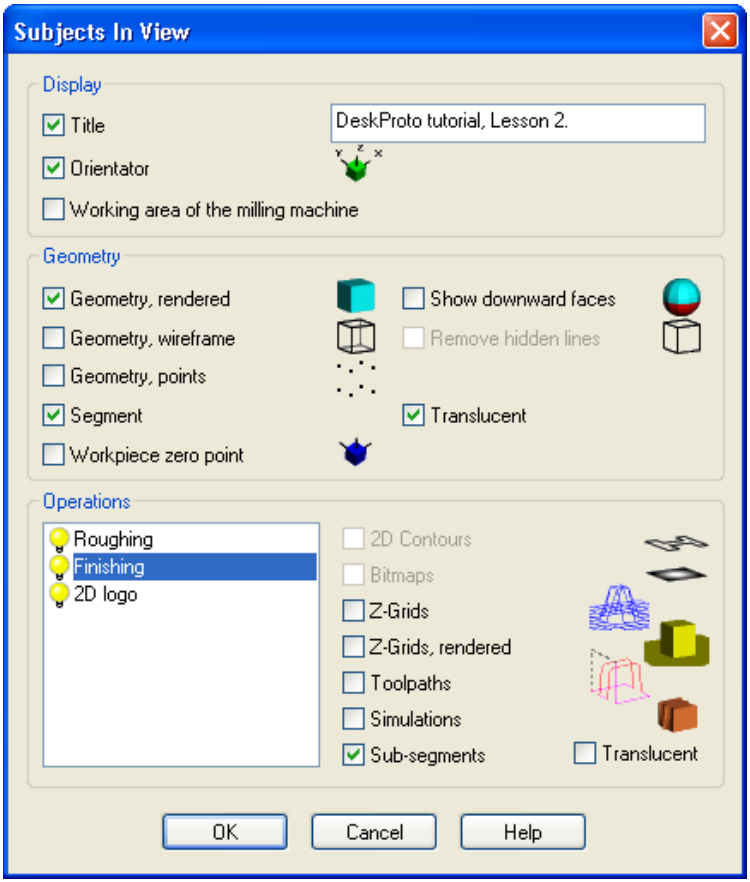
The View menu command Subjects... displays the **Subjects in View dialog,** in which you can change what is and what is not shown in the active view (the Scene). In case none of the boxes is checked the View Window will be blank. You can also select which operations should be visible.

Shortcuts

Toolbar: 

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

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



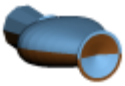
The **Title** will be displayed on the screen and on the printed view, in the top left corner of the View.

The **Orientator** is the coordinate system icon (cube with axes) displayed in the bottom left corner of each view. It helps you to understand from which direction you are looking at the geometry. This green cube does NOT indicate the zero point.

The **Working Area** can be drawn, which is the working area of the machine selected for this part. Of course DeskProto does not know where you will set the workpiece zero point, so it will draw the part exactly in the middle of the working area. This will give a good indication how the part relates to the machine. The working area for your machine can be set in the Machine dialog.

The second group shows the Geometry-related subjects:

In a **Rendered geometry** drawing all triangles of the geometry definition are made "solid" with color. This is the default Subject in View as this offers a good understanding of your geometry.

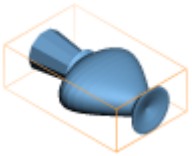


The sub-option **Show downward faces** will assign a different color to any triangles that are facing down (of which the normal has a negative Z-component). This option makes it very easy to check for Undercuts, and to optimally rotate your geometry to reduce undercuts.

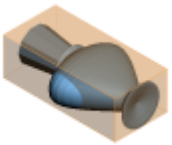
In a **Wireframe geometry** the triangles are drawn with lines.
it is possible that you can't see the geometry very well because all the lines are confusing.

In such case the 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. Of course these are only visible in case Wireframe geometry has not been not selected.



The **Segment** shows the rectangular bounding box of the part in light brown lines. This is in fact the block of material that is needed to machine your part. Note that often only the green lines of the sub-segment will be visible, when both segments are equal.



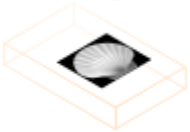
The sub-option **Translucent** draws the sides of the segment in a translucent color (for the Part segment in brown). This will shown the segment block more clearly, maintaining visibility of the geometry inside.

A second **Orientator** is available at request, drawn exactly at the workpiece zero point (in blue). The position of the Workpiece zero point can be changed using the Translation options of the Part Parameters. Note that the small black dot on the screen is NOT the zero point: it indicates the first point of the toolpath.

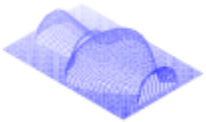
The selection of subjects in the third group may be different per Operation. It is for instance possible to show the toolpaths of Operation Roughing and not show them for Operation finishing, while both operations are visible.

In the list of **Operations** you can change the visibility of each Operation by clicking on the lamp icons. Yellow (lamp on) means visible, gray (lamp off) means invisible. A red lamp indicates an error status for that Operation. This is an alternative for clicking the lamp icons in the Project Tree.
In the list you can also *Select* one or more Operations: making the line blue (meaning selected) by clicking on it. Two or more operations can be selected by keeping the Control or the Shift button depressed when clicking. The Subjects that you will check and uncheck for the Operations, apply ONLY to the selected operations.

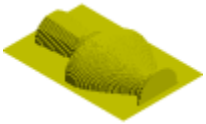
2D Contours concern the 2D drawing(s) that are used in 2D Operations. This option is available only when one or more 2D Operations are selected.



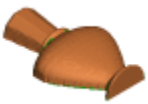
Bitmaps concern the Bitmap drawing(s) that are used in Bitmap Operations. This option is available only when one or more Bitmap Operations are selected.



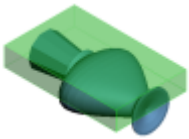
When you check the **Z-Grids** checkbox the Z-grids of the visible operations will be shown. The Z-grid is an intermediate representation of the geometry that DeskProto uses for it's toolpath calculations. The Z-grid will be drawn in lines.
The same subject will be show for option **Rendered Z-Grids**. Now the Z-grid is drawn as a rendering, so you can clearly see that it is a 3D bar graph representation of the geometry.



When you check the **Toolpaths** checkbox the Toolpaths of the visible operations will be shown. The toolpath that is drawn is the same toolpath that will be sent to the machine: if there are any problems it should be possible to detect them now.



When you check the **Simulations** checkbox the Simulations of the visible operations will be shown. You can clearly see what the resulting model will look like.



When checked a **Sub-segment** will be shown: the rectangular bounding box of the operation in light green lines. The sub-option **Translucent** draws the sides of the segment in a translucent green color. This will shown the segment block more clearly, maintaining visibility of the geometry inside.

The colors that are used for the various subjects in the example drawings above are the DeskProto default colors. These can be changed in the Colors tab of the DeskProto Preferences.
For all line drawings default fog may be applied to simulate depth, which can be set on the same location.

VII Various

7.1 Initial Personal Settings dialog

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 you create and for every part you add to your project. This setting is stored in the default Part, and can be changed later by choosing the option Default Part Parameters from the Options menu.

You also have to set the measurement unit you want to use within DeskProto. This unit will be saved in the Preferences. You can also change this setting later, by choosing the Preferences dialog (Options menu).

7.2 Template Project

Sometimes you deal with a series of projects that very much look like each other: all settings equal, however with slightly different geometries. In that case you don't want to define the parameters for each project from the start.

A first solution is to then enter all parameters as **Defaults** (default Part and default Operation). When you then start DeskProto and load a geometry, 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. This solution only is useful 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 without a geometry, which is called a **template project**.
In DeskProto you can set most parameters without loading a geometry. When saving such project DeskProto will warn you that no geometry is present: in order to create a template project continue on that point.

Here some examples:

- You might want to define a project-template you can use for a certain material. For instance when you always machine dental models your settings are always the same. However some models are in zirconium and some in wax, needing different settings.
- You might want to define a project-template for moulds, using inverse milling.
- You might want to define a project-template with one part defined as top part, another defined as bottom part (with correct transformations), each containing one operation for roughing, and one for finishing.

To use such a template project, open the project which serves as a template, and load the geometry file that you want to use. Save it directly after loading under a different name. After that you can safely edit the new project without changing the template project.

A third option for automation is writing a Script.

7.3 Command Line Parameters

It is possible to run DeskProto with a command line parameter. This is not so much useful for 'normal' users, but rather for application builders who need to include toolpath calculations in their application. So when you do not know what a command line parameter is please ignore this paragraph.
Other ways to more or less automate DeskProto are Template projects and Scripts.

The easiest Command line parameter is to use the name of the **geometry file** (STL file). In the Run dialog of the Windows Start button you can for instance enter the following command:
"C:\Program Files\DeskProto 5.0\dp.exe" casting.stl
This will start DeskProto and automatically load the Casting geometry. The STL file needs to be present in the Data directory as set in the DeskProto Preferences.

You can also specify the STL file including it's complete path, like in:
"C:\Program Files\DeskProto 5.0\dp.exe" "C:\User\STL Data\medallion.stl"

It is also possible to automatically load a **DeskProto project file** (DPJ file) on start. For instance:
"C:\Program Files\DeskProto 5.0\dp.exe" torso.dpj
This project may also be a Template project.

It is also possible to automatically load a **DeskProto script wizard file** (DPW file) on start. For instance:
"C:\Program Files\DeskProto 5.0\dp.exe" "C:\Program Files\DeskProto 5.0\Wizard\123waxRing.dpw"

It is also possible to automatically load a **DeskProto script file** (JS or VBS file) on start. For instance:
"C:\Program Files\DeskProto 5.0\dp.exe" "..\Scripts\ScriptedBottle.js"

A very convenient way to configure a command line parameter is by editing the Shortcut on the desktop (click it with the right mouse-button, in the context menu choose Properties, select Tab Shortcut): you can add the command line parameter to field Target.

The above options start DeskProto and load data, nothing more. Calculating toolpaths and writing the NC file still have 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 5.0\dp.exe" casting.stl 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 parameters used will be the parameters as set in the defaults. This works great when the application involves repeating the same job with a slightly different geometry. The application programmer can include DeskProto in his application without bothering the end-user. Calling DeskProto with the correct command line parameters can be done by a macro that runs in the CAD software used, or by a stand alone application.

In this case too: the first file may be either an STL file or 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 STL file used has the same name for all models.
This option allows you to achieve full automation like mentioned above, for two sided milling.
For instance:
"C:\Program Files\DeskProto 5.0\dp.exe" TwoSided.dpj TwoSided.nc
When you do this with a template project file DeskProto will prompt the user to browse an STL file

Both command line parameters can include 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.

This trick also works in case the parameters cause DeskProto to save more than one NC program file 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, also extending the NC filename using the standard DeskProto naming conventions for multiple NC files.

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.

Finally it is also possible to load a **Script file** (JS file or VBS file) and a **Script Wizard file** (DPW file) as first command line parameter, making DeskProto automatically start that script / that wizard.

7.4 Scripts and Script Wizards

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 either in **VBScript** (Visual Basic Script, file extension .vbs) or in **JScript** (JAVA Script, file extension . js).

A script can be called in DeskProto using the command **Run Script...** in the File menu.
You can also start a Script by calling it as command line parameter.
Default directory for script files is the Scripts subdirectory of DeskProto (C: \Program Files\DeskProto 5.0\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 to the Scripting Interface. Most of these objects are Properties (a variable that can be set or read) and Methods (a function with one or more parameters).

A very simple example of a DeskProto script file is present already (ScriptedBottle.js):

```
DeskProto.project.loadGeometry(false, "C:\\Program Files\\DeskProto 5.0\\Data\\Bottle.stl");
DeskProto.project.activePart.setRotation(-90.0, 0.0, 0.0);
DeskProto.project.activePart.segmentMethod = 2;
DeskProto.project.calculateToolpaths();
DeskProto.project.writeNCProgram( "C:\\Program Files\\DeskProto 5.0\\Data\\ScriptOutputNCfile.ext" );
DeskProto.exit();
```

This script will load the Bottle geometry, orientate it correctly for machining, set the segment to use the upper half only, calculate the toolpaths using the default Operation parameters and save these toolpaths in an NC program file.

The available scripting objects are described in the **DeskProto Script Reference**: a series of HTML files that can be found in the directory \DPScript\ on the DeskProto CD. Open the file \DPScript\index.html to start reading.

A special type of Script that is available in DeskProto is the **Script Wizard**. This is a very powerful option, making it possible to add you own Wizards to DeskProto. Such wizard can make it easy to use a special machine, or to create a special type of product. A good example are the Wax Ring Wizards

that come with DeskProto:

- the **123WaxRing Wizard** in combination with the 123WaxRing fixture make it very easy to machine a wax ring from three sides.
- the **5thAxisWaxRing Wizard** makes it possible to use the 5t axis of the Roland JWX-10 milling machine to create hollow rings.

Both wizards are Script Wizards, and can be found in C:\Program Files \DeskProto 5.0\Wizard. You are free to change these (if you know what you are doing) or use these as examples for your own script wizards.

A DeskProto Script wizard is a series of HTML files, each file containing one script to be executed. Each of these HTML files defines one page of the Wizard, and you can use any HTML option to layout this page.
For more information see the DeskProto Script Reference just mentioned.

All Script Wizards can be started via **Start Wizard** in the File menu: the combo box at "Other" shows all available script wizards.
You can also start a Script Wizard by calling it as command line parameter.

7.5 Graphically finding the rotation

Finding the correct XYZ rotation values to orientate the geometry as required can be very difficult, especially when angles other than 90 degrees are involved. Here is a tip to easily find these three rotation values by first rotating the geometry on screen.
Follow the following steps:

1. Set the rotation settings for the Part to [X=0, Y=0, Z=0].
(Part Parameters, tab page Transform)

2. Rotate the geometry to the required orientation, using the mouse, 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 help that you can use here is the option called Show downward faces in the Subjects dialog.

7.6 Compatible CAD Software

As DeskProto supports STL files, the most widespread de facto standard file type used by CAD-systems for prototype building, file transfer should be possible without many problems. Any current 3D CAD system can export STL files. In addition, the two formats DXF and VRML that are available as well guarantee a successful geometry transfer from nearly any source.

For 2D data the formats STL and VRML do not apply, so only DXF is supported. To create 2D data typically a different type of software is used: 2D drawing software, instead of 3D CAD software. Many 3D CAD packages do include some 2D options as well, though.

We do not offer a complete listing of compatible software (as this would be too long for this manual), however a few remarks on export filters for a few widespread CAD systems are in order.
Note: all trademarks are owned by their respective owners.

General remarks:

The original Stereolithography system has more severe requirements for the **STL file** than DeskProto. It accepts only positive coordinate values, small gaps of say 0.001 mm between two triangles are lethal, a complete and true solid must be present. DeskProto does not care about all these errors.

In some solid modelers, for instance in Pro Engineer, extra care is needed to export STL files as they require a solid model for this option. For stereolithography this is needed indeed, and it might cost a lot of extra work to get a valid solid. For DeskProto the STL file need not be a valid solid.
Both binary and ASCII STL files are accepted.

Some surface modelers produce STL files that contain invalid normals. In that case the drawing in DeskProto shows part of the surface in black. The reason for this is that for part of the geometry the normal-vectors in the STL file are reversed (resulting in an invalid solid). You can correct this by switching off the option 'Skip Backfaces' in the Edit Project parameters dialog box.

For **DXF files** the situation is a bit more complex. Almost any CAD system can export DXF, however 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.
In general it is advised to use STL, however for some CAD packages STL is not supported and DXF should be used instead:
3D Studio Max (older versions)
Corel Dream 3D

Form Z (older versions)

The same applies to **2D DXF** files: here too DeskProto supports only a subset of the complete file definition. The supported entities are: Point: will result in a drilling operation on that location.
Line: will result in a tool movement from begin to end (linear interpolation). Polyline and LW polyline: a series of lines and/or arcs connected to each other. Arc: will be converted to many small line segments by DeskProto.
Circle and Ellipse: will be converted to many small line segments by DeskProto.
Any Z-coordinate values in the file will be ignored by DeskProto.

The standard for **VRML files** is almost completely covered by DeskProto, both for VRML Version 1 and Version 2. Not supported (yet) are the entities Sphere, Cone and Cylinder, and the concept of Custom Node Type definitions. Note that the file-extension for VRML is WRL.

A file type that is also used to transfer data between CAD and CAM systems is the **IGES file**. DeskProto does not support IGES files, as they store the data in a much more complex way, making all types of conversion errors possible. In theory IGES offers higher quality transfer than STL, however for prototyping purposes STL is sufficient. IGES may have advantages for moldmaking, as more tooling strategies are possible on IGES data. Many high-end CAM packages are offering this type of functionality, but this at high cost both in money and ease-of-use.

Program specific remarks:

Old versions of AutoCAD behave strangely for 3D objects:
In **AutoCAD R11/R12** an STL file can be generated using the command SOLSTLOUT, which is a part of the AME modeling extension. The geometry must first be placed in the positive octant (all X, Y and Z values positive).

In **AutoCAD R13** the command STLOUT must be used, which is only possible for 3D solids (geometry created using the solid modeling option). First move the part to the positive octant. In case you are using the Designer parametric extension you first have to explode the part before running STLOUT. You can then Undo to get back to the part as originally designed.
The accuracy can be controlled using the AutoCAD system variable FACETRES. Its range is between 0.01 and 10, the default value is 1, which is OK for most parts. Increasing this value will increase the number of facets (and the STL file size as well).

In Autodesk **Mechanical Desktop** you also have to build a solid before you can write an STL file. Here the procedure is to first explode the solid (only the part, not 'all'; remove it from any assembly first), then set the facet

resolution (advised: max = 10) by typing "facetres <0.5>:10", move the solid in the positive X, Y, Z octant, and then do file...export...file type STL.
Use a continuous line layer for this entire procedure.

Newer AutoCAD versions and **Autodesk Inventor** just show STL as one of the available export options.

JewelCad is a CAD program special for jewelers, and it combines very well with DeskProto for the creation of wax models. There is an issue with the dongles of Jewelcad and DeskProto, for which a solution is available. In case you use these programs and have a dongle problem, please contact Delft Spline Systems for support

In older versions of **Pro/Engineer** the required export option is not called STL, but SLA (Stereo Lithography Apparatus) instead. Pro/E offers two parameters to set the accuracy of the STL file: Chord Height and Angle Control. Do note that a true solid is needed to enable this option.
For Chord Height enter 0 (zero): Pro/E then will show the minimum and maximum permitted values. Use the minimum value for a smooth prototype.
For Angle Control in most cases the default of 0.5 is OK. For smoother surfaces use a higher value. In case you are asked if non-negative coordinates are needed, both options are okay for DeskProto.

The **Rhino** software has a special link with DeskProto that is already built into the program. When both Rhino and DeskProto are installed on the same PC, then DeskProto can be called directly from Rhino using the command “ Print 3D” in the Rhino File menu. Rhino will than start DeskProto with your current geometry loaded.
When creating an STL file in Rhino you have many options to set the accuracy, so many that it may be confusing. The Preview buttons offers a good help here, as the preview offers a good indication if the STL file will be sufficiently smooth.

In **Solid Edge** STL output is only possible in the Part editor, so not for assemblies. If this is still needed you can convert an assembly (ASM file) to a part using the 'Insert' command in the Part editor. As the object to insert you can choose an assembly, which will then be converted to a part and after that can be exported to STL.

In **Unigraphics** two parameters are used to define the accuracy of the STL file: Triangle Tolerance (tri tol) and Adjacency Tolerance.
Triangle Tolerance must be set between 0.0025 and 2.5 (mm). The default of 0.8 results in a poor part: use a value around 0.01 instead.
Adjacency Tolerance is only needed for surfaces, to prevent small gaps between the triangles. Use the default (this setting does not influence DeskProto).
Note: in Unigraphics, STL export is only available as optional extra, at an

extra price.

In fact most 3D CAD packages do offer a few parameters to set the accuracy of the STL file (and thus the number of triangles and the file size). Many will also offer some 'Preview' option, giving you feedback about the values to use for an acceptable resolution.

Index

D

E

F

G

H

I

J

K

L

M

N

O

P

Q

R

S

T

U

V

W

Z