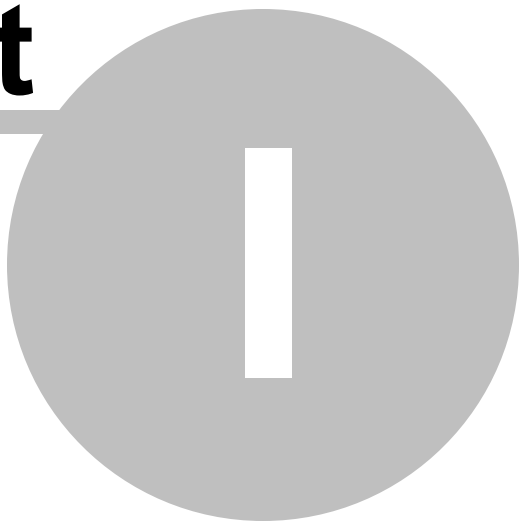
cadXtract

Table of Contents

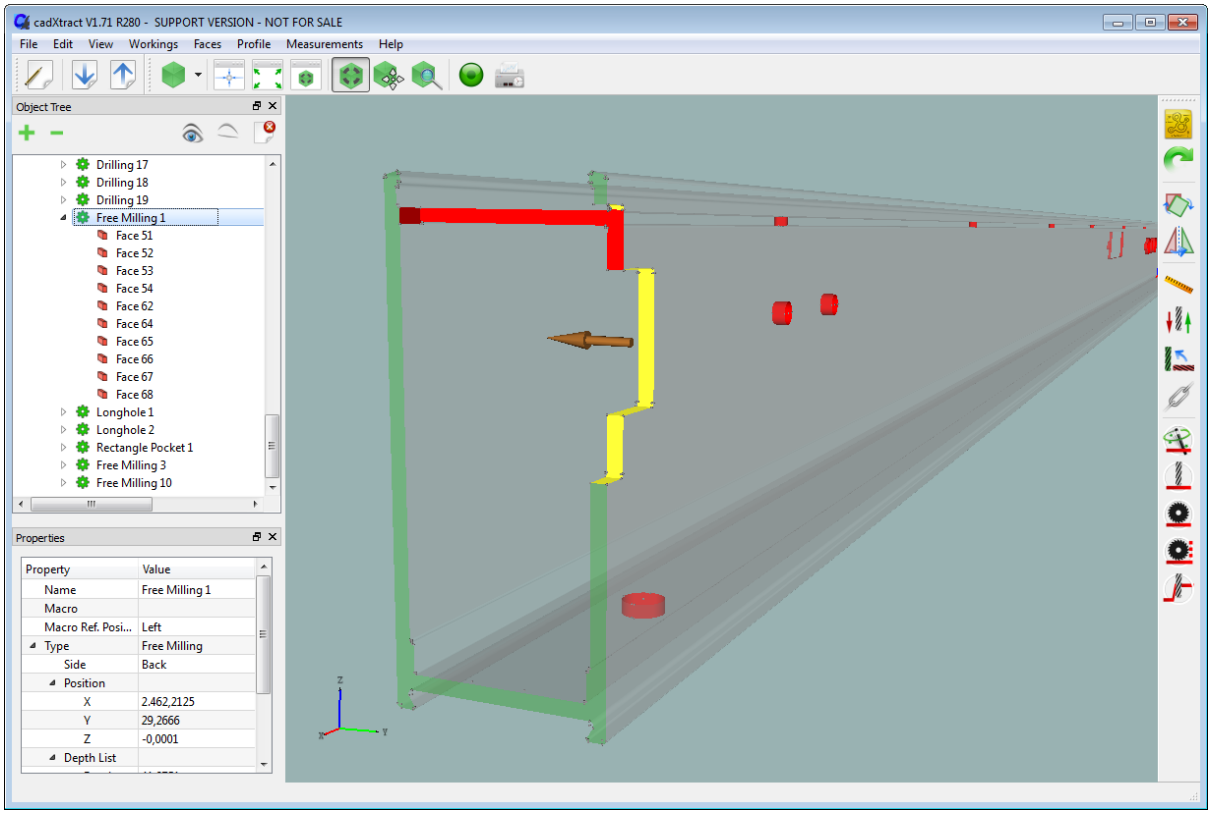
Part



1 Introduction

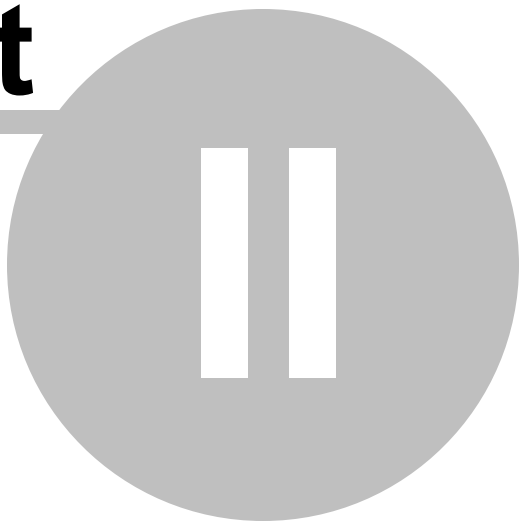
cadXtract reads your 3D-data, analyzing the models autonomously and creates NCX-data automatically.

Background: as in 3D-models you only find information about the planes of a working piece (no more data about their development – the feature information), an automatic data transfer is in general very complicated and often needs an extensive manual post-edit process (pick the relevant lines or points, define the intrusion direction etc)



cadXtract independently scans the “planes of interest”, analyzes them and calculates all necessary data for the operations. These data are exported after this to the CAM-software.

Part

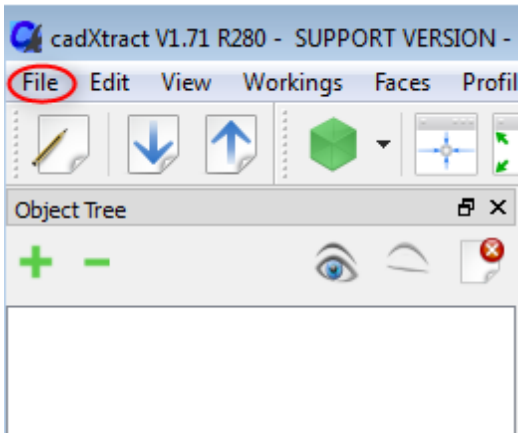


2 General

As cadXtract is a nearly simple converter from STEP-files (3D-models) into ncx-files, the most important functions are

"Import" and "Export"

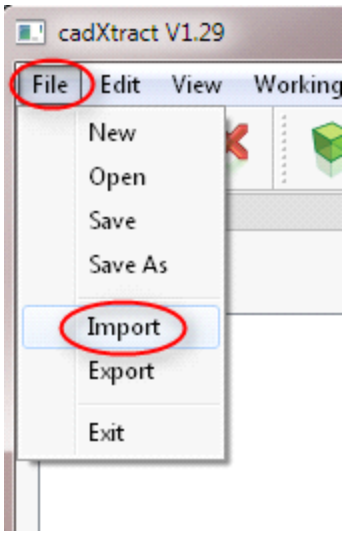
You find them certainly in the menu "File"



Rem : You can set the default - directory in the Preferences

<u>The General way to use cadXtract is :</u>

1) Import a STEP-file
2) (if necessary) check and rework the detected operations
3) define (if the profile shape is not detected before) the profile-ident-number
4) export the NCX



Import starts the following process :

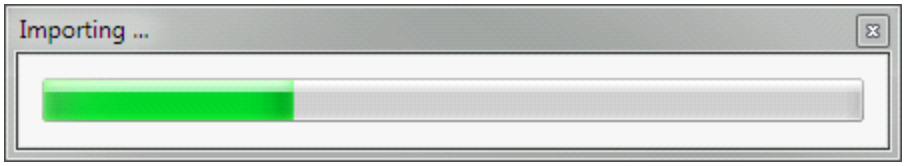
1) reading the STEP-file : a STEP-File contains only information about all faces in the construction and a lot of data administrative data

2) sorting the faces into
- profile / raw material - faces
- operation - faces (these faces do not belong to the profile)

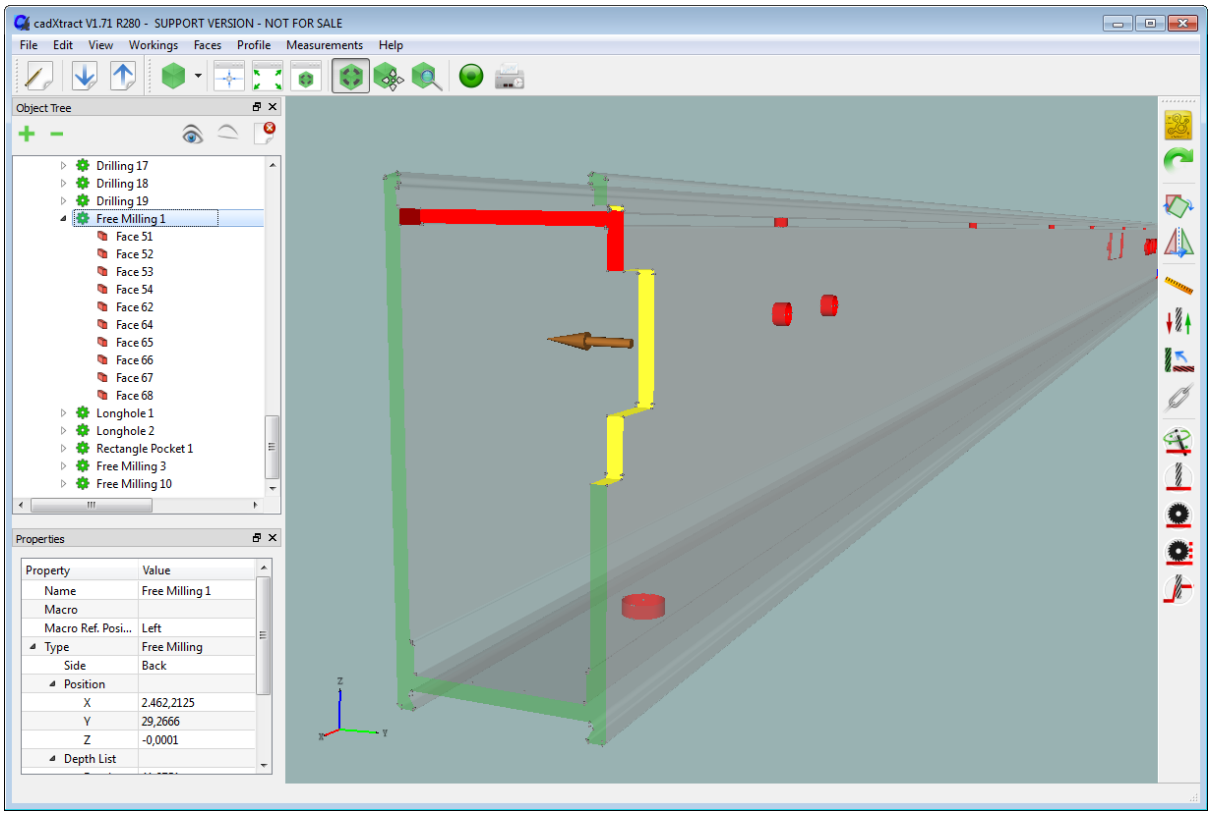
3) analyzing the faces
- connected faces are interpreted as an operation
- the definition of these operations is taken out from the face-information

4) operations, which are not detected as standard will be marked as an "Unknown operation", in this case the operator has to "help" the converter by picking one or more faces and start the detection with this selection again.

While this caculations are done, the operator sees the following window ...



Now you will get a screen similar to this one :



In some cases you find "Unknown operations" (as described), which you have to control.

"Unknown operations" results out of one of this cases :

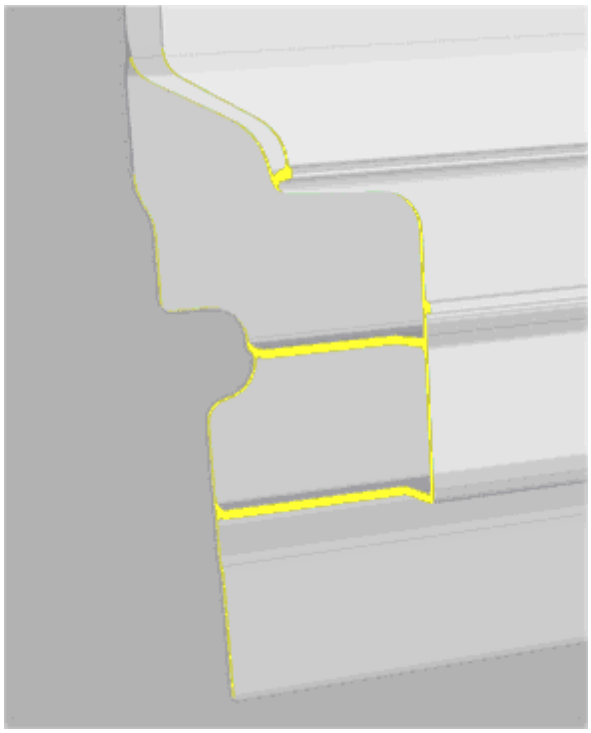
1) there's only 1 face for this operation : this cannot be analyzed automatically, because 1 face can be handled as

- a milling contour (from alt least 2 directions)
- a milling contour from the orthogonal direction
- a sawing cut

In this case you have to select one of these possibilities

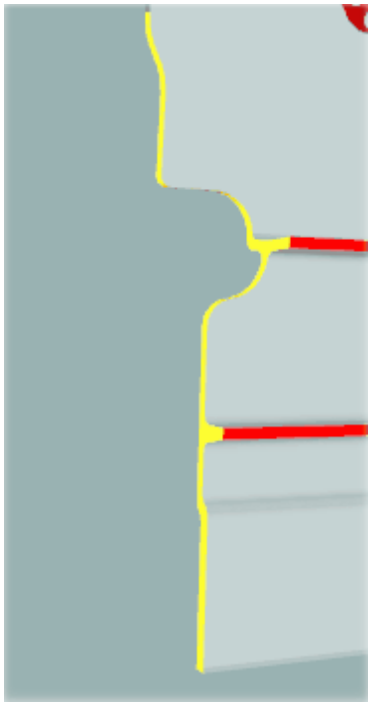
2) the ensemble of the faces which are connected have not one collective "intrusion vector", to wit this are more than on operation from different sides (e.g. top and front). In this case you have to select the faces (1st the faces which refers to the operation from the top and 2nd the faces for the operation from the front side) and start the detection - in genera
l "Detect Working"

3) the ensemble of the faces does not lead to a proper 2D-contour,



In this case you have to select the 1st contour, start "Detect working" and the same for the 2nd contour

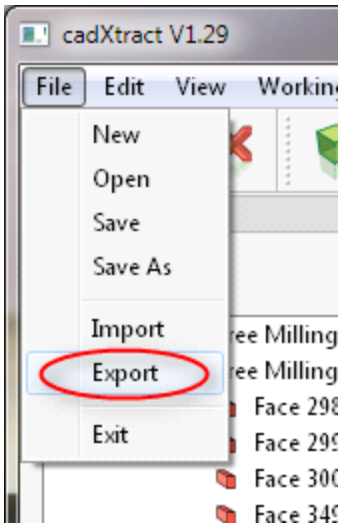
1.contour



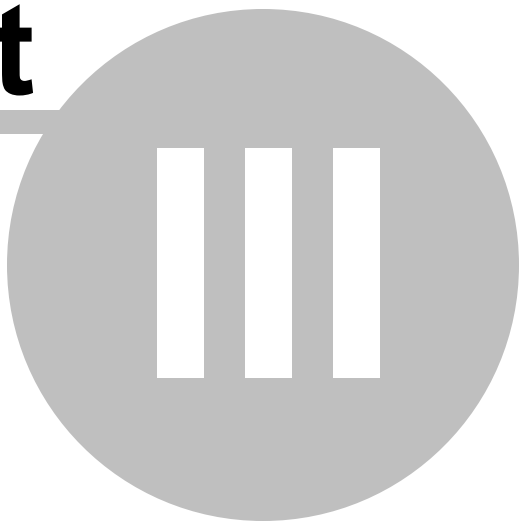
2.contour

4) He faces contains B- or C-Splines : these are 3D-planes, which would refer to 5-axis-interpolation. Neither cadXtract nor camQuix are designed for free modeling. cadXtract is able to read the outer contour of these shapes ("Detect Working As Outer Shell Milling") to reduce the information to a milling from the front or from the back.

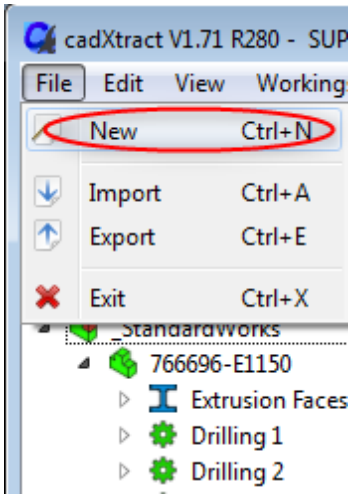
5) To produce the part, you have to export it as a NCX-fil to a CAM-system (camQuix)



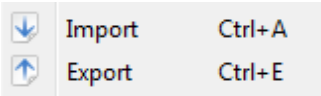
Part



3 File



With the function "New" you can create an empty project.
Important in this menu are :



Import

To start reading and analyzing,
(Please see details at General)

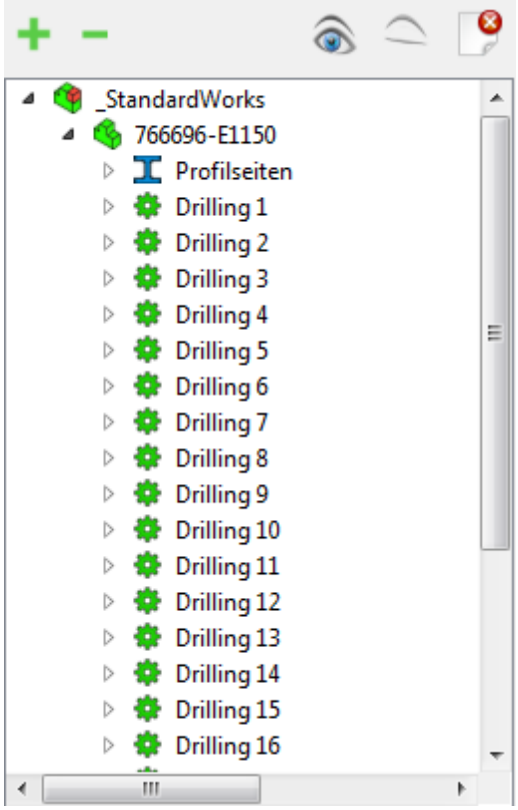
Export

To write the data into a NCX-file

The list of defined operations will be written into a NCX-file.
At the same time the profile shape is exported to a cc-file,
so you can load the ncx-file in camQuix
and you will see directly the correct profile shape in the correct situation.

If you have problems that the operations does not fit to the profile in camQuix, please check:

- the Profile directory (--> Preferences) is set correctly (...\temp)
- the directory and the cc-file are not write-protected
- camQuix does NOT refer to a DXF-file, which defines another profile or another situation

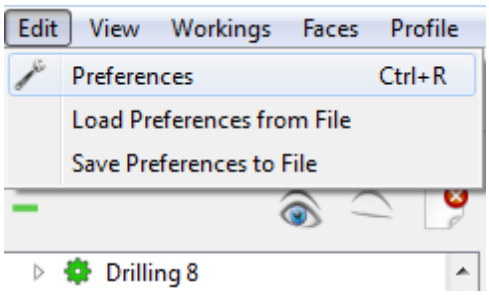


Out of the operation-list all operations are written as they are defined (independent of the number of assigned faces). Only "Unknown Operations" are NOT exported.

Part IV

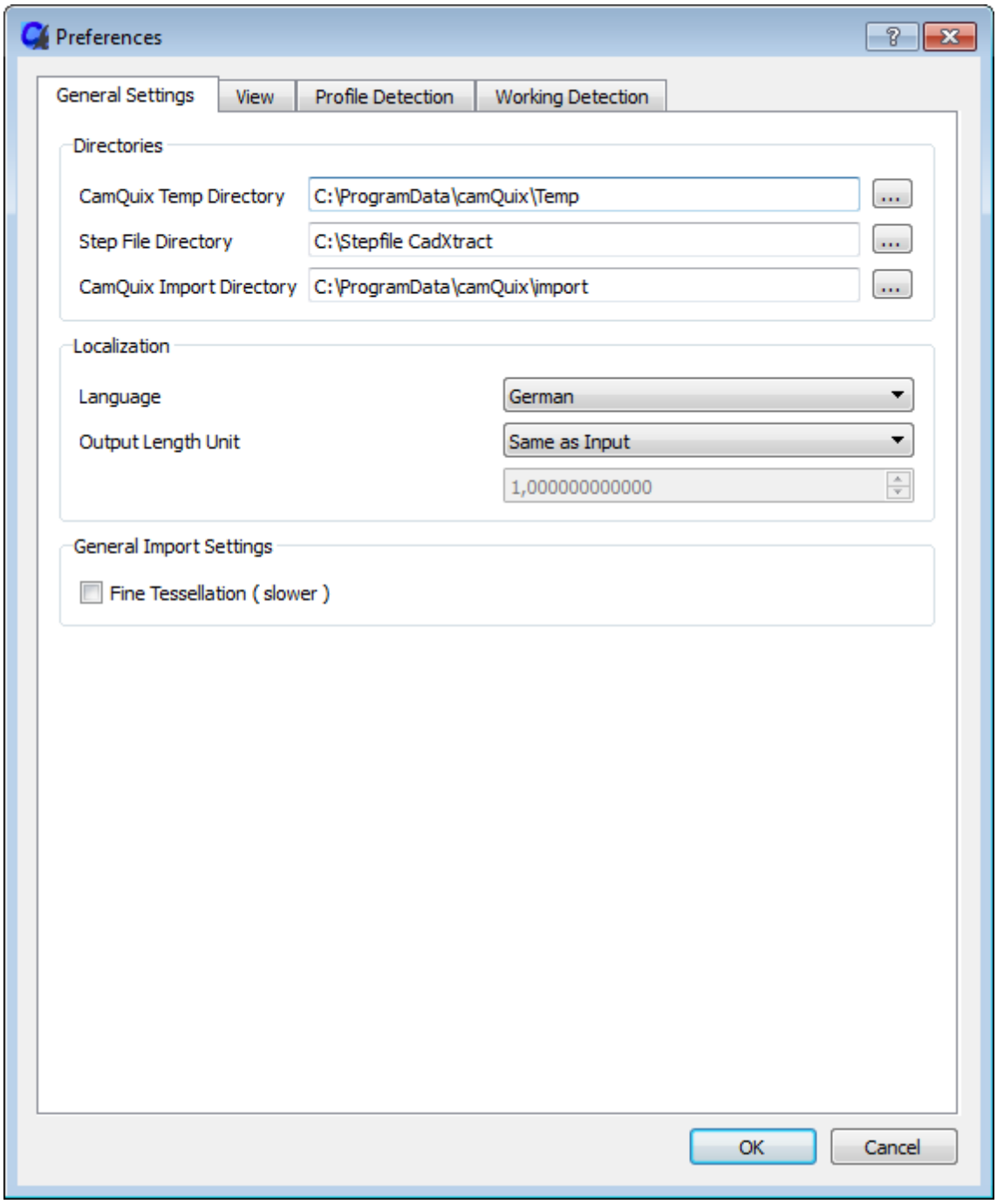
4 Edit (Preferences)

Here you can do the settings, where cadXtract looks for the Step-files (the default import directory), where it writes the ncx-files and different other settings :

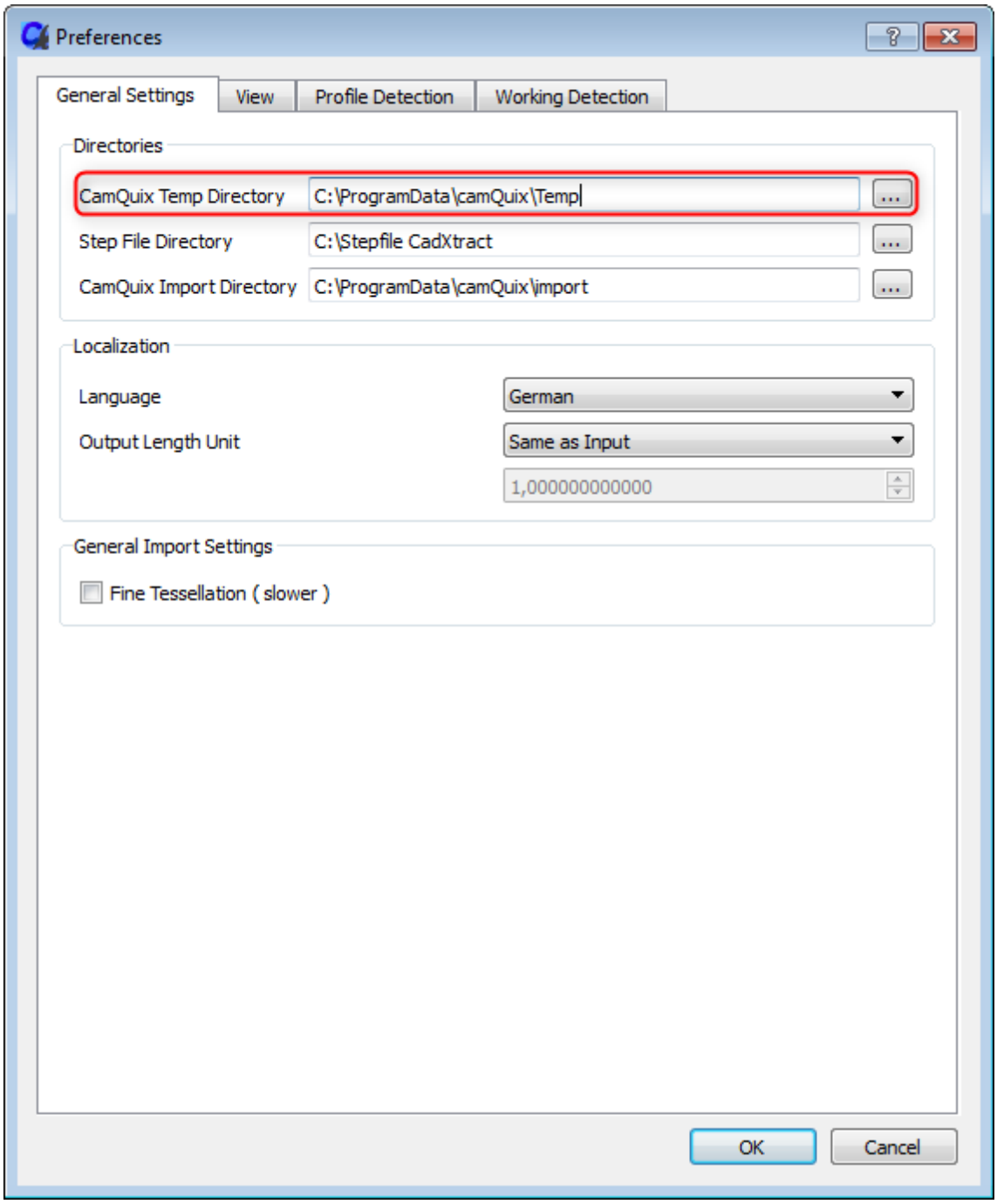


So you can change the options in the following window :

General Settings :



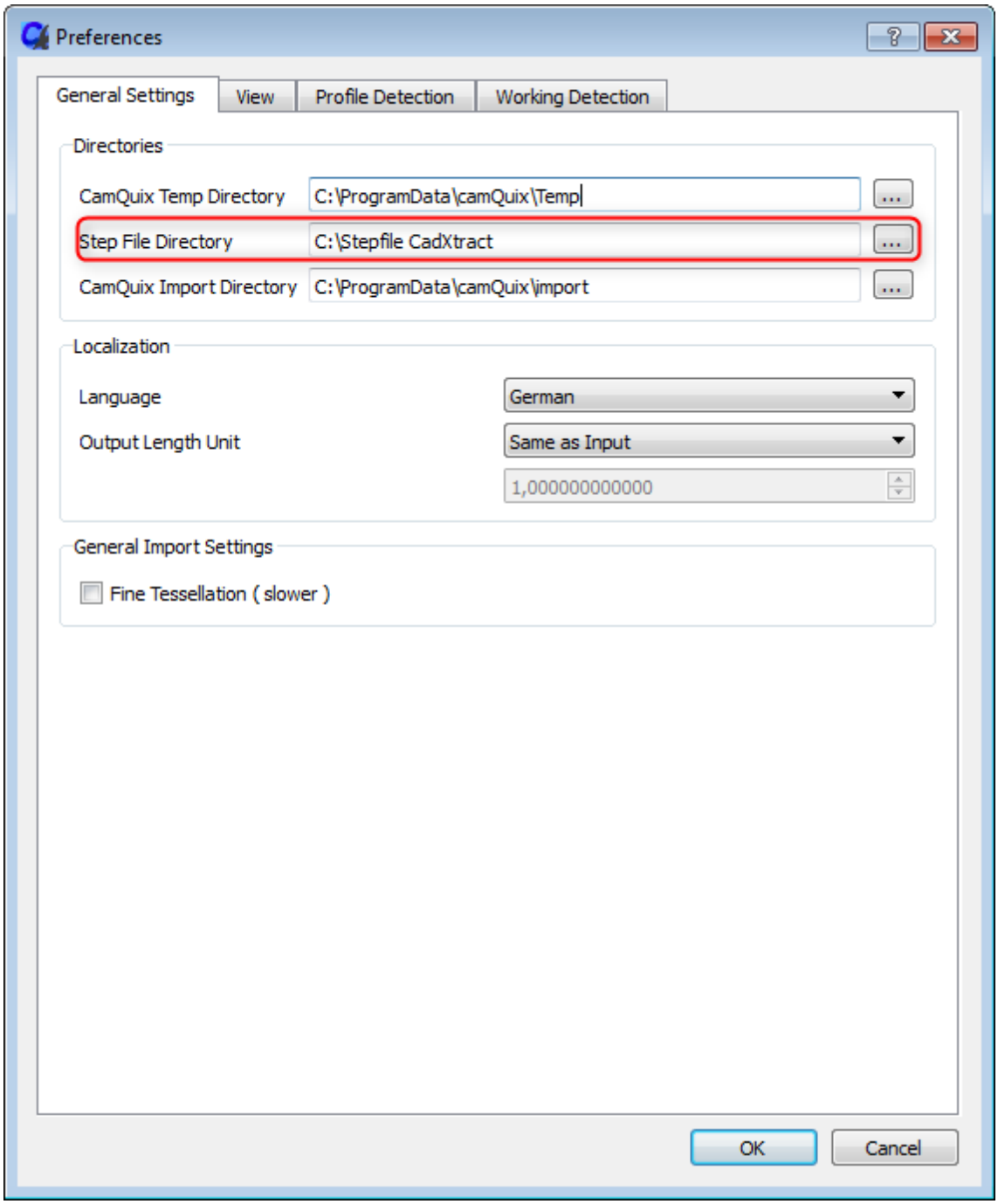
camQuix Temp Directory :



This is the path, where cadXtract writes the .cc-files, that means the cross-cuttings of the detected profiles.
To optimize the speed of cadXtract they are exported directly in the camQuix-own format to enhance the speed.

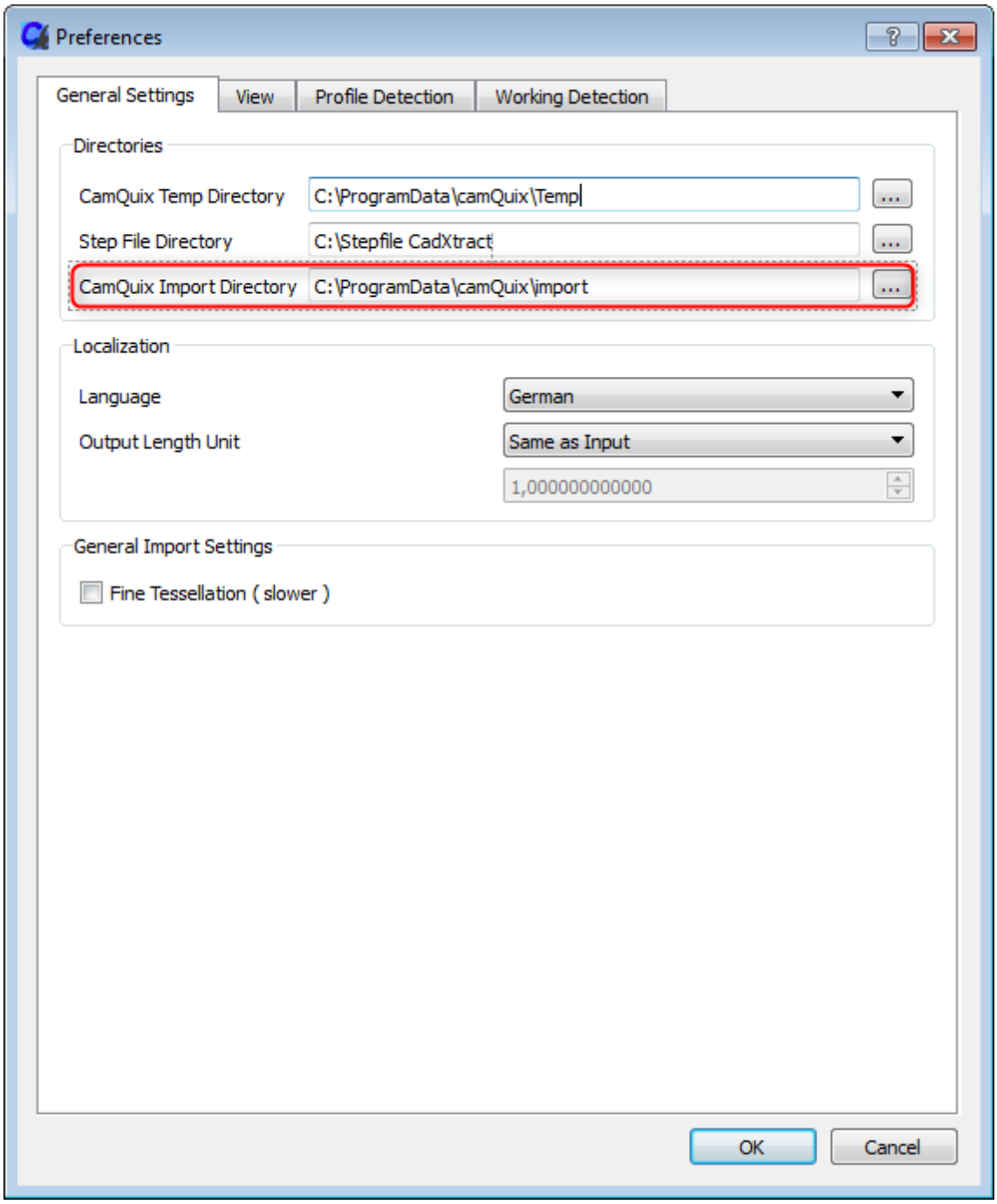
If you are working with camQuix you have to set this path to the "TEMP"-path of your camQuix-installation !

Step File Directory :



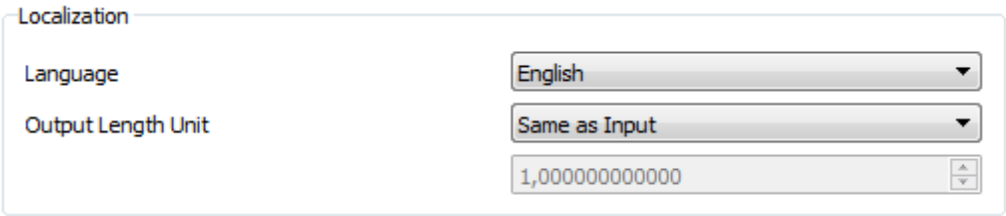
This is the default-path, where cadXtract looks for the STEP-files. Certainly you can change later the directory...

camQuix Import Directory :



This is the default export-path, where cadXtract writes the ncx-files. You can define it finally when you are exporting the NCX-file,
but it is useful to set this path near by the camQuix-import-path.

- Localization :

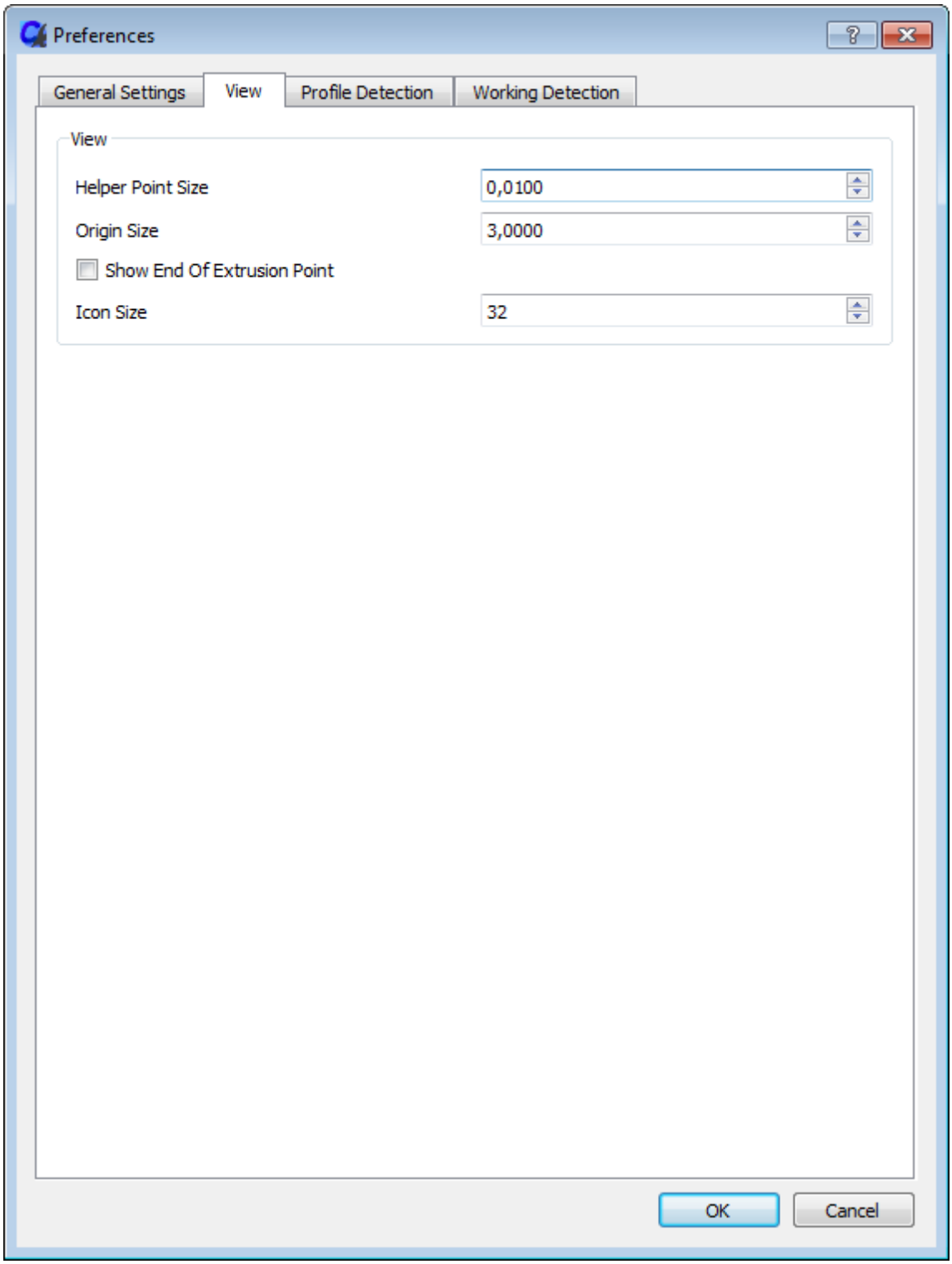


Here you can choose the language you want and also choose the output length unit like "mm" "inch" or "free", so that the output file becomes another unit.
Please notice that the database for already detected macros will not be updated if this is changed so you should not change this setting if you have an existing database.

- Fine Tessellation:

If you choose this option, the tessellation will be finer, that means the triangles that will be calculated for the models will be smaller
but the import speed of the choosen file will be due to that slower.

View:



In this window you can change the following options:

- Helper Point Size :

Define, how big the hepler points will be (the little grey dots in the 3D model)

They will be e.g. needed for measurements. This can be done by pushing the button  .
For further informations take a look at "Symbols".

- Origin Size :

Here you can choose the factor of the size of the origin helper point.

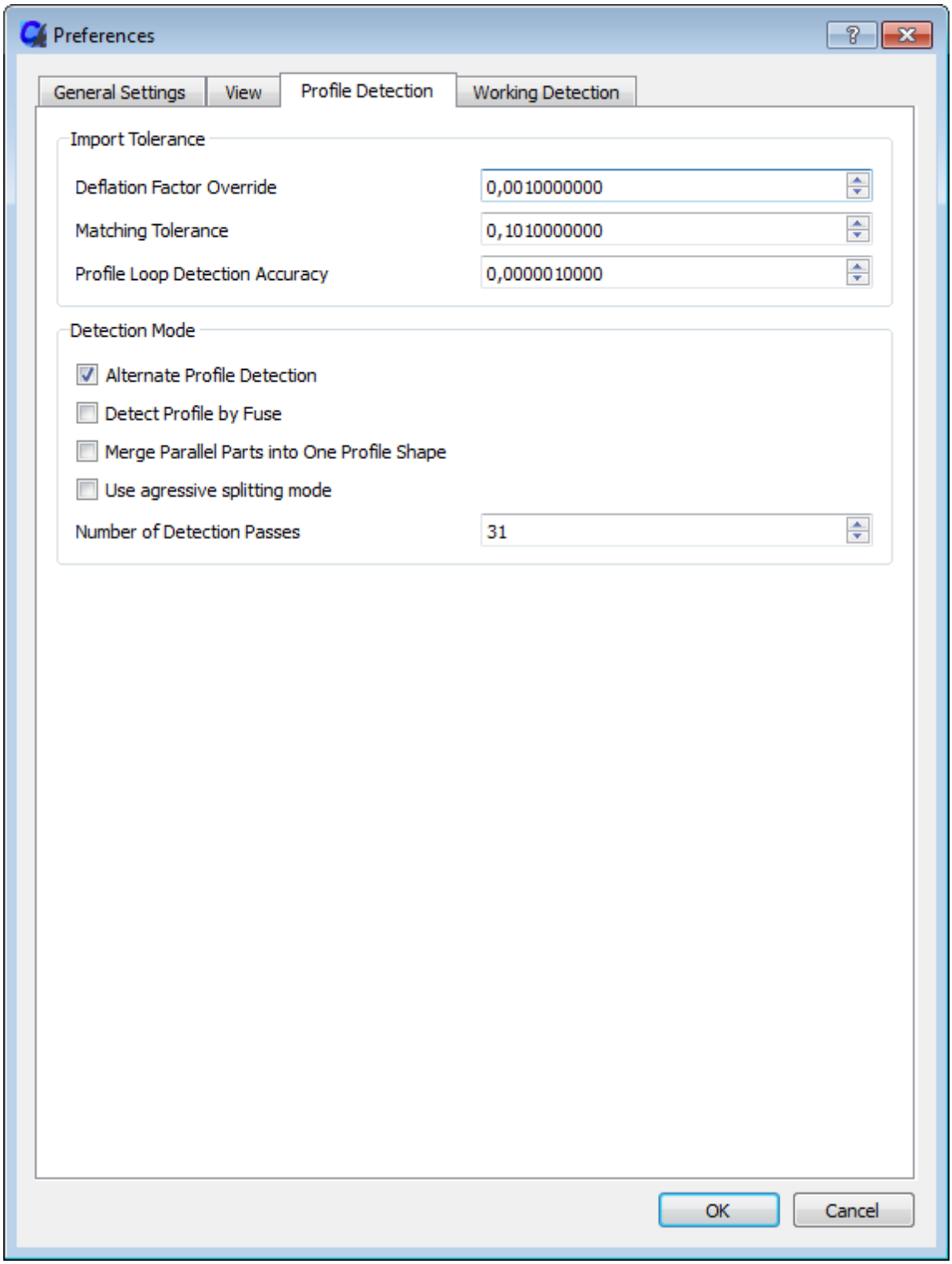
- Show End of Extrusion Point:

This option is used more for control. It shows the theoretical endpoints of the drawing if it is cut by compound angles.

- Icon Size:

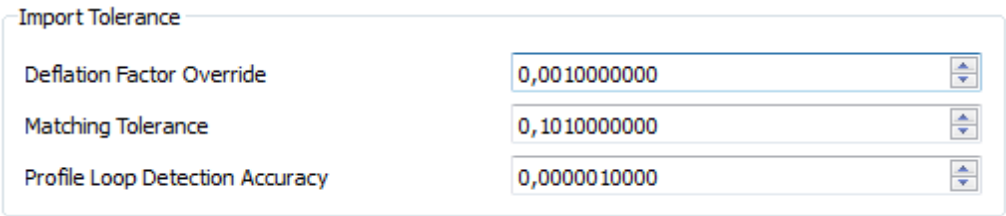
Choose how big the icons will be.

Profile Detection :



In this window you can affect how the profile you import will be detected

Import Tolerance:



- Deflaction Factor Override:

Accuracy for arc interpolation.

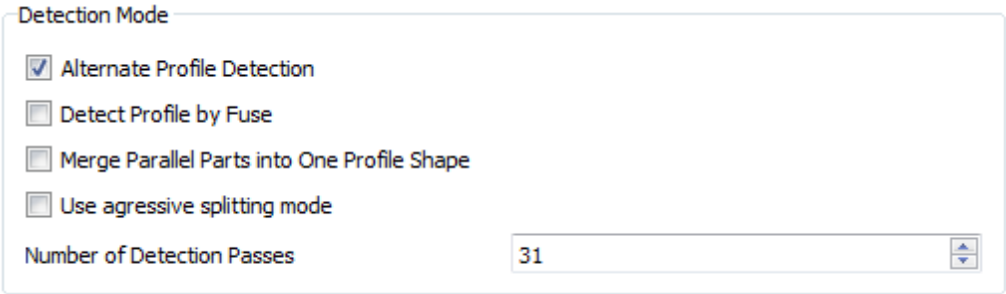
- Matching Tolerance:

Accuracy used for profile matching .

- Profile Loop Detection Accuracy:

Accuracy used for profile loop detection.

<u>Detection Mode :</u>



- Alternate Profile Detection:

This option might fix issues for holes in the profile detection.

- Detect Profile by Fuse:

Use this to detect a lot of big workings in the profile.

- Merge Parallel Parts into One Profile Shape:

Parallel extrusion profiles are merged into one profile. Can be useful if you profile contains seals.

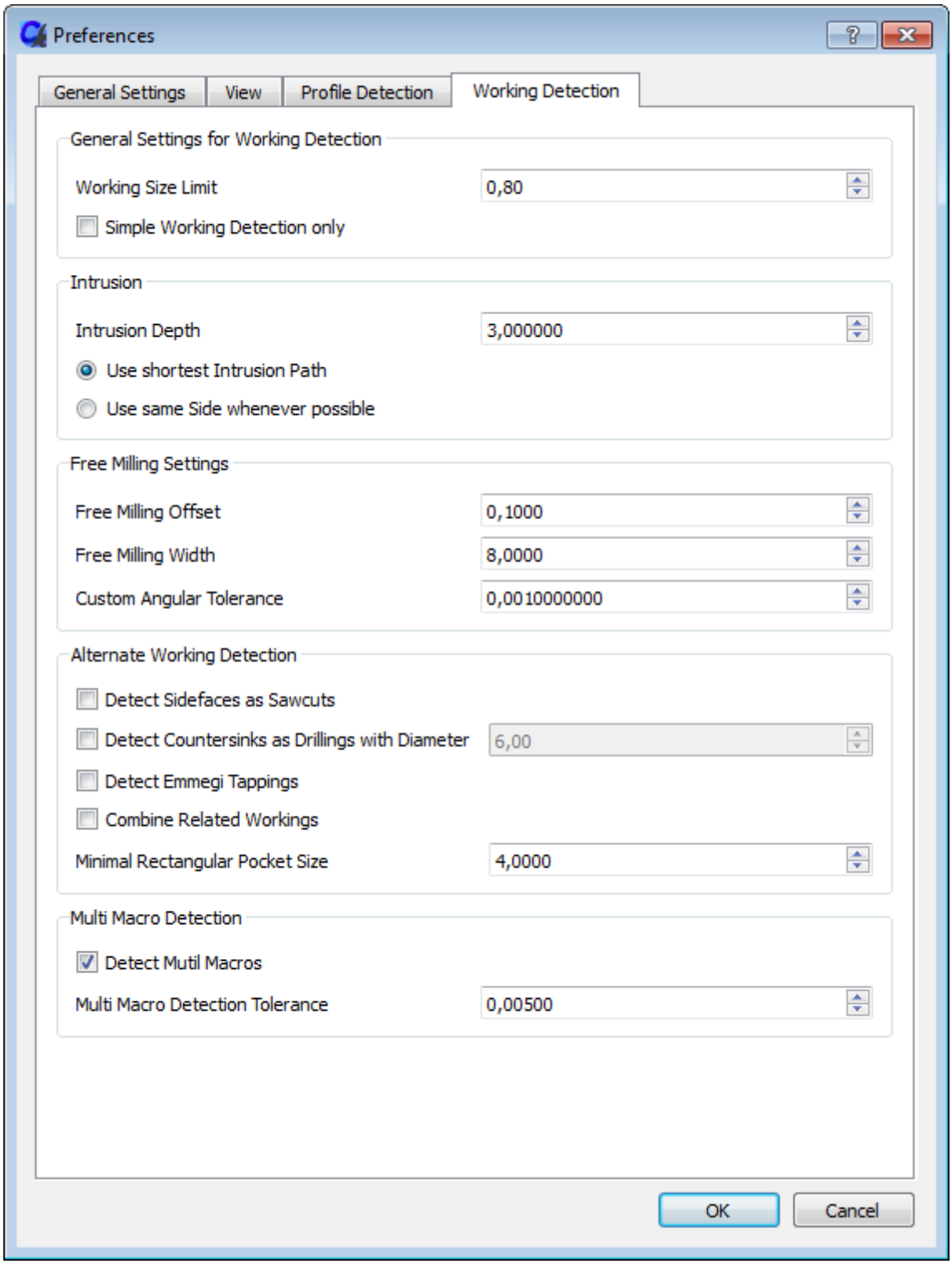
- Use agressive splitting mode:

Try really hard to split faces that are part of the profile as well as part of a working (most of the time a free milling).
Waring: This is extremely slow.

- Number of Detection Passes:

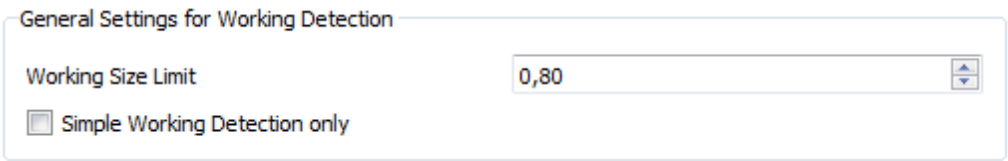
Adjust the number of passes to detect the cross section of the profile.
If your profile only contains a few workings you might reduce this to gain speed.
If you recognize profile cross-section-detection issues increasing this value might fix this.

Working Detection:



The options here will take affect on how the workings will be detected of the file you import

General Settings for Working Detection :



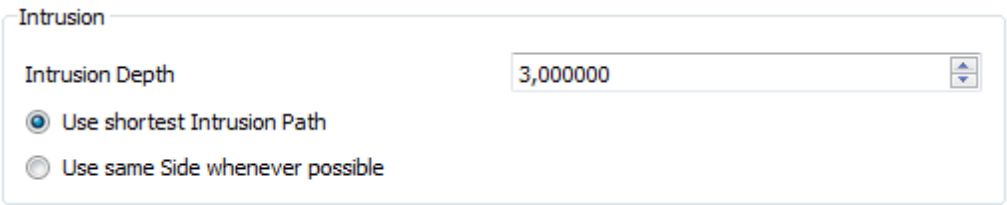
- Working Size Limit:

This is the maximum size of a working in relation to the profile's length.

- Simple Working Detection only:

This skips the automatic detection of free milling, tappings and more advanced notchings.
Can be very useful if you have a lot of very complicated workings in you part that willl likely be unknown workings.

Intrusion :



- Intrusion Depth:

Default 3,0.

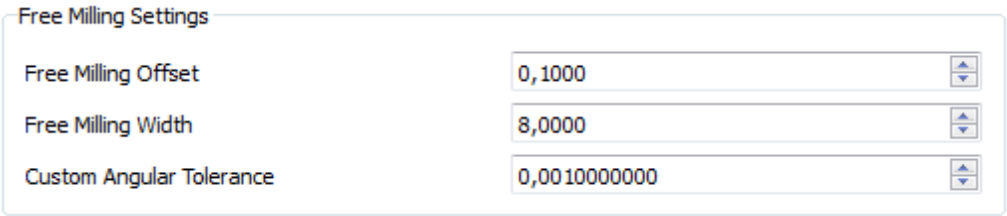
- Use shortest Intrusion Path:

Try to use the side for the intrusion that has the shortest intrusion path.

- Use same Side whenever possible:

Try to create all workings on the top/front/left side if possible.

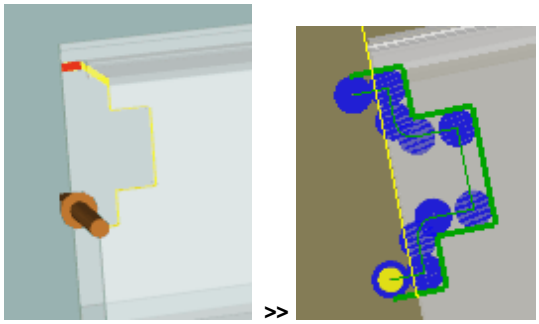
Free Milling Settings:



- Free Milling Offset:

Adjust the free milling offset. This is the distance from the tool to profile at start.

When cadXtract detects a free milling, it generates the contour (if it starts at a corner of the profile) a"little bit" longer,
so the tool does not have to enter half in the profile - the intrusion point is shifted outrside of the material:



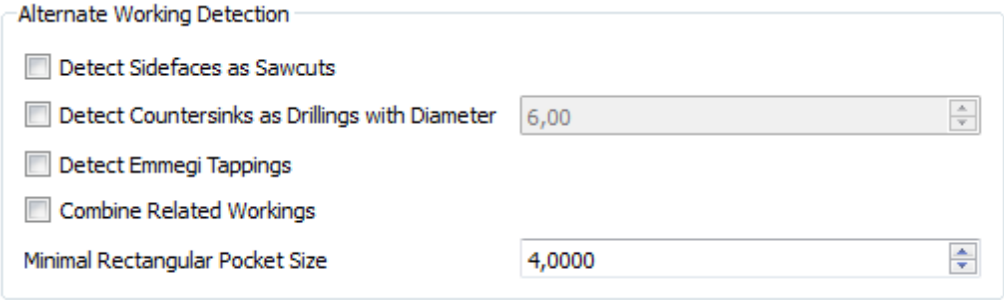
- Free Milling Width:

This is the standard width of the milling shapes in the exported ncx-file (This leads to the milling diameter of the assigned tool)

- Custom Angular Tolerance:

This option is very useful, if the models are designed a little bit improper.
This gives camQuix the possibility to handle the intrusion vector with a tolerance,
that means also workings, which are not exactly in the same direction (+-0.1°) will be sorted together with a dept table.

<u>Alternate Working Detection:</u>



- Detect Sidefaces as Sawcuts:

The sidefaces of the profile are detected as basic saw-cuts.
This makes it easier to work with the CAM Software.

-Detect Countersinks as Drillings with Diameter:

All found countersinks are transformed into drillings.

-Detect Emmegi Tappings:

Emmegi uses a special geometry to model tappings.
They are only recognized if this option is true.

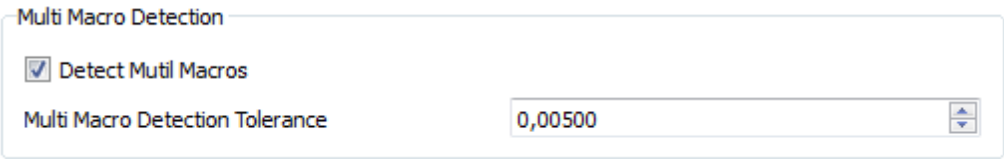
-Combine Related Workings:

Combine workings that relate to each other.

-Minimal Rectangular Pocket Size:

Minimal size that is used to detect a single plane as rectangular pocket.
Smaller faces will generate free millings (chamfers)

Multi Macro Detection:



- Detect Multi Macros

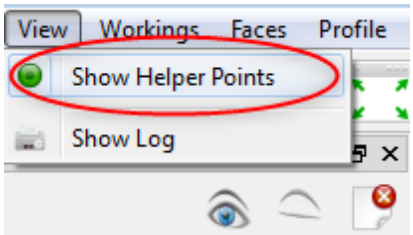
Let the software detect multi macros in the profile.

- Multi Macro Detection Tolerance

Set this to zero to use the countersinks inner diameter for the drilling

Part V

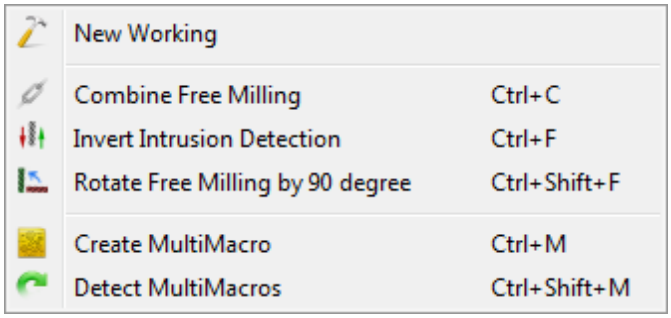
5 View



Please notice that when the helper points are shown that that the view on details may be blocked.

Part VI

6 Workings



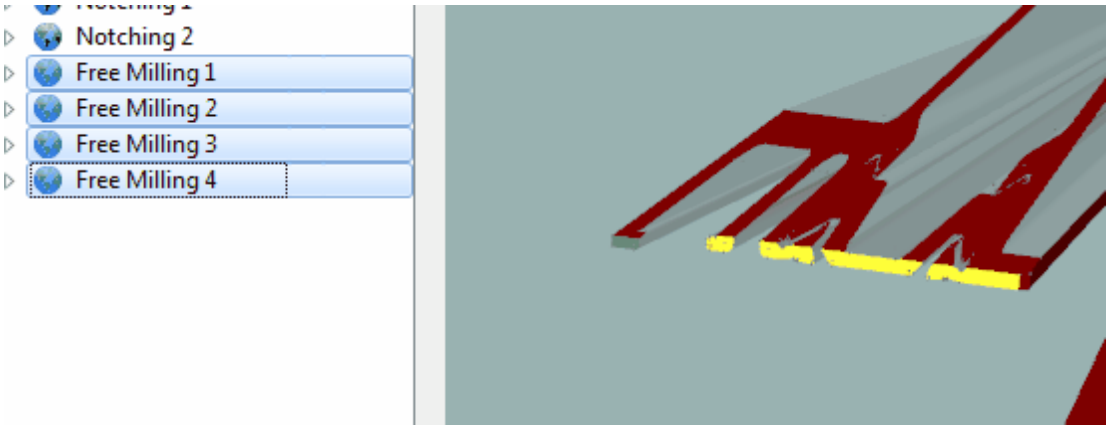
New working:

If necessary you can add manual defined operations here. This makes sense, if you want to pick some "difficult" positions or dimensions out of the model. Else it is easier to define additional operations in camQuix.

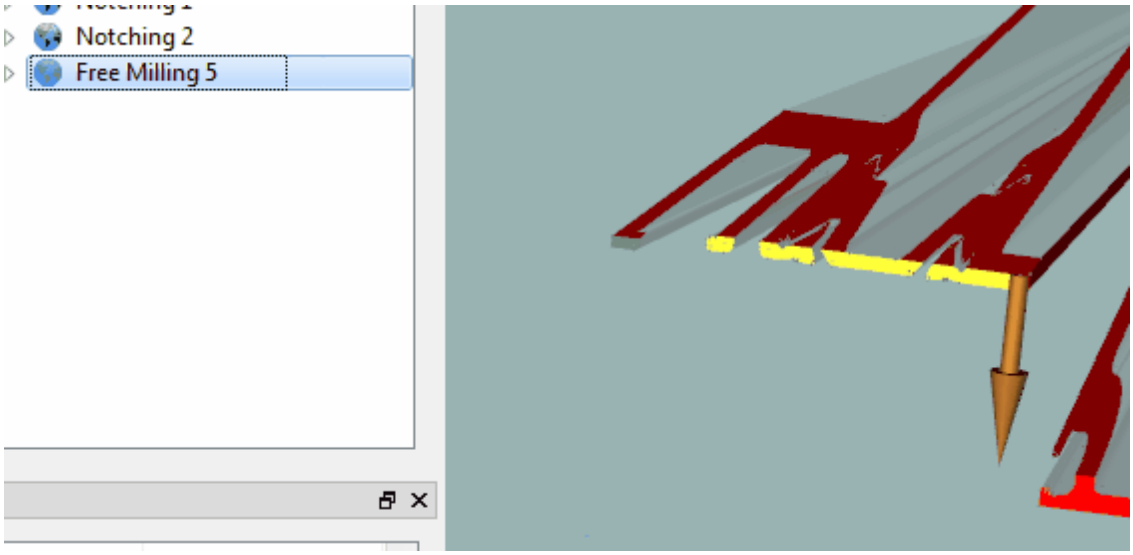
Combine free millings:

If a model contains some separate free millings which should be exported in one milling (because of the profile contour), you can combine them.

Example : this 4 millings are detected separately (which is correct)



In a real production every operator would have programmed manually a ONE contour. To tell camQuix to combine them into one contour, you have to select the 4 millings and combine them



So they will be exported by one ...

Invert Intrusion Detection:

Invert the intrusion direction of the current working.

Rotate Free Milling by 90 degree:

Rotate one-faced free millings by 90°

Create MultiMacro:

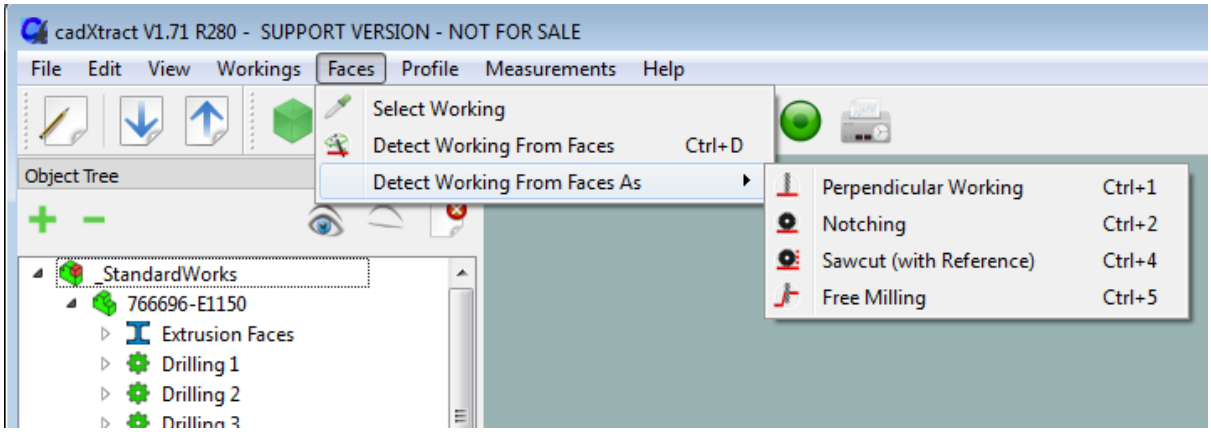
This option is an extra module and only available if paid for.
With this you can put several workings into a so called MultiMacro.

Detect MulitMacro:

Same condition as for option before.
With this the program is able to detect MultiMacros.

Part VII

7 Faces



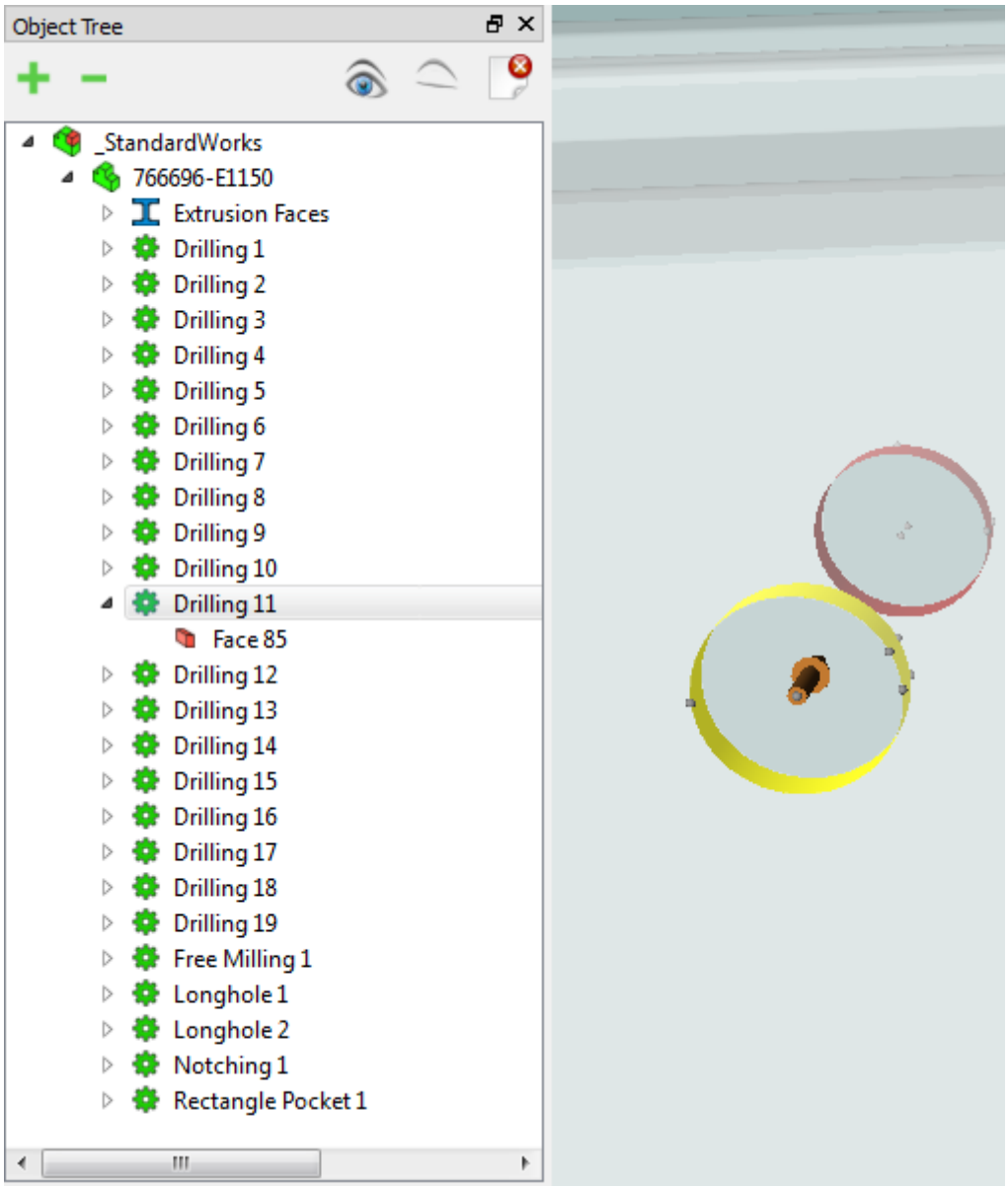
Select Working

You can move the actual selection of faces to an operation (which you have to choose than). It is easier to move select faces to an operation by drag and drop them on the operation line in the list-view.

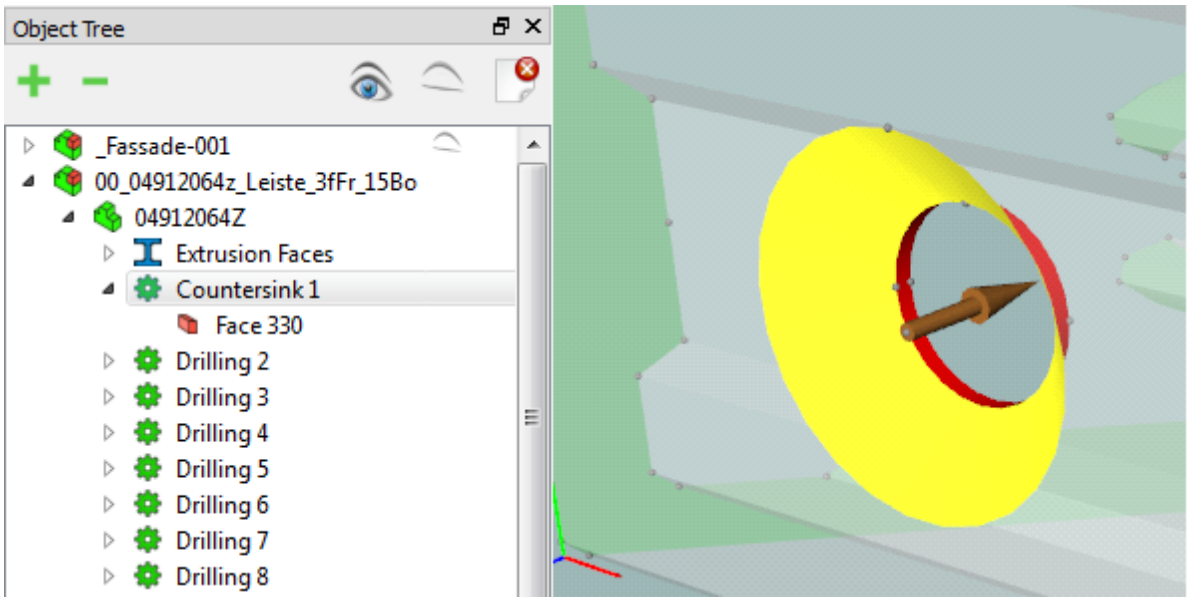
Detect Working

This is the most used function, if you want to rework a model.
As cadXtract works as automatically as possible, you will need this feature to detect workings out from a manual selected number of faces (As described in General).
cadXtract tries to detect a standard operation out of the selected faces :

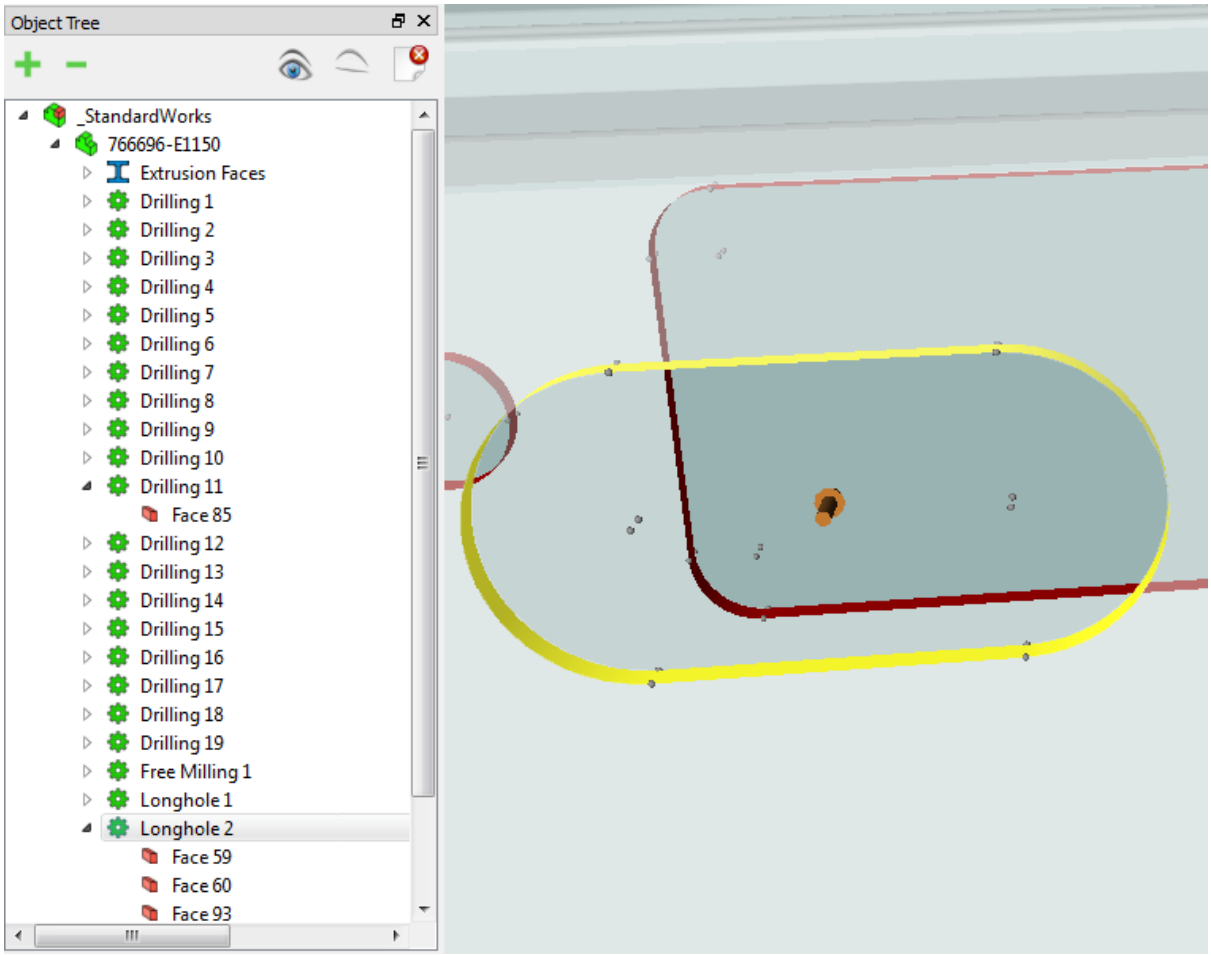
Drilling:



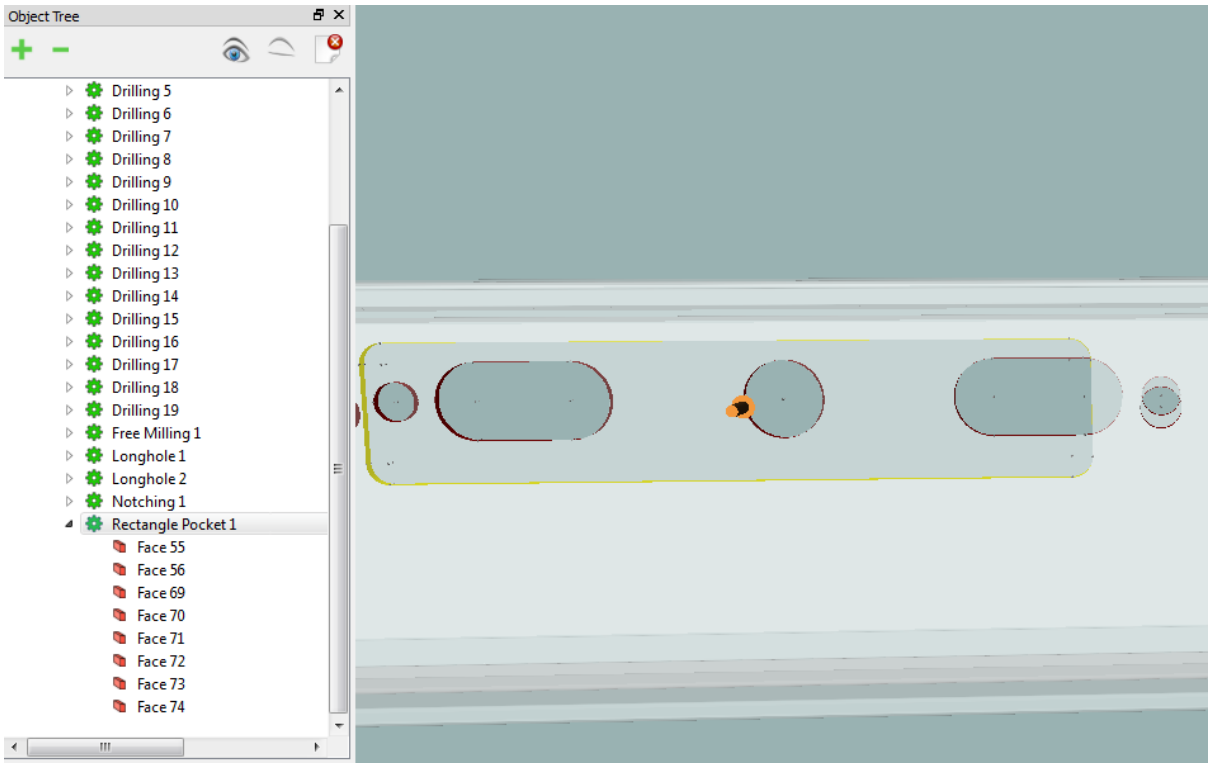
Countersink:



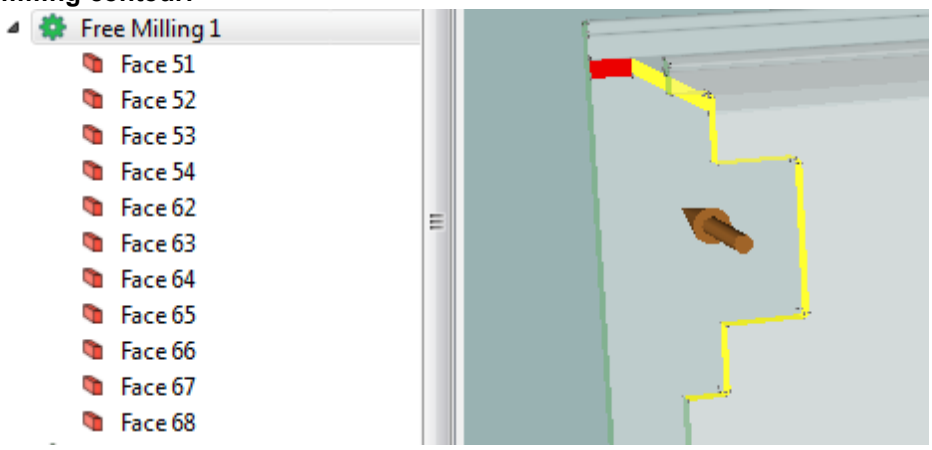
Slot:



Rectangle:



Free milling contour:

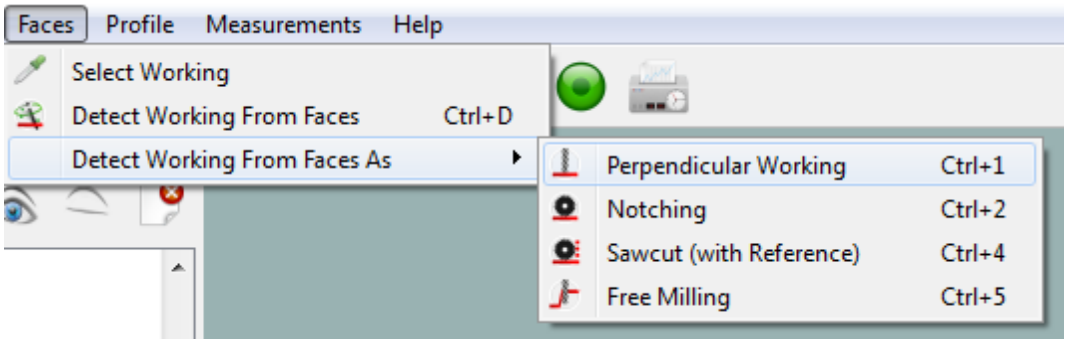


Detect Working As ... "Perpendicular Working"

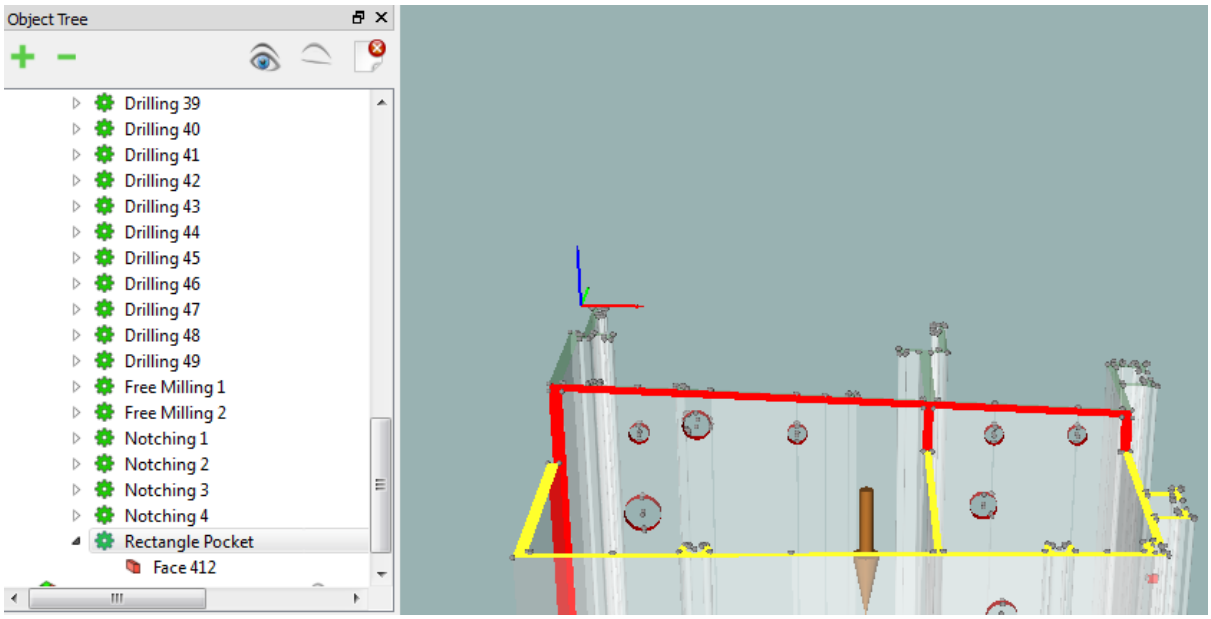
This is used often used to generate a bevel : an orthogonal milling.

Further it can be used to generate a rectangle pocket, which is orthogonal to the picked face :

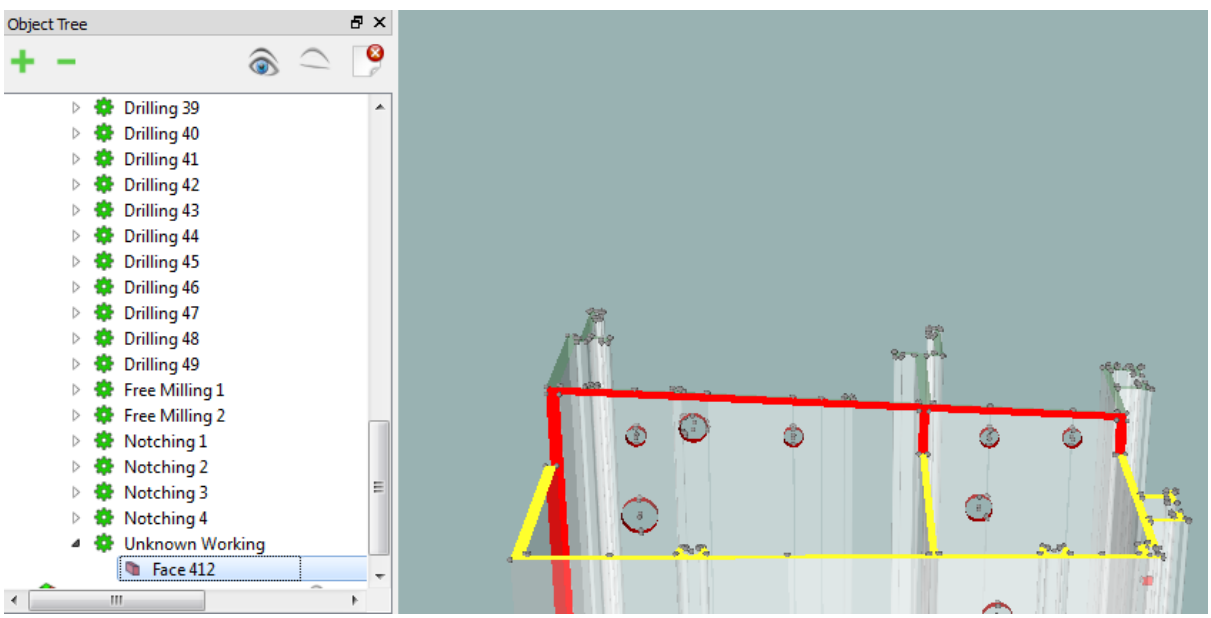

Object Tree
Drilling 39
Drilling 40
Drilling 41
Drilling 42
Drilling 43
Drilling 44
Drilling 45
Drilling 46
Drilling 47
Drilling 48
Drilling 49
Free Milling 1
Free Milling 2
Notching 1
Notching 2
Notching 3
Notching 4
Unknown Working
Face 412


Faces
Profile
Measurements
Help
Select Working
Detect Working From Faces Ctrl+D
Detect Working From Faces As
Perpendicular Working Ctrl+1
Notching Ctrl+2
Sawcut (with Reference) Ctrl+4
Free Milling Ctrl+5

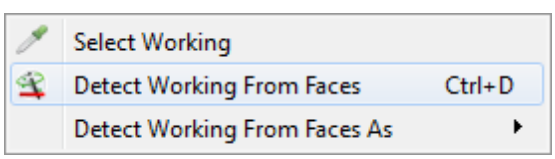
leads to...



(normally this face will be detected as a free milling from the side (please see the arrow !)



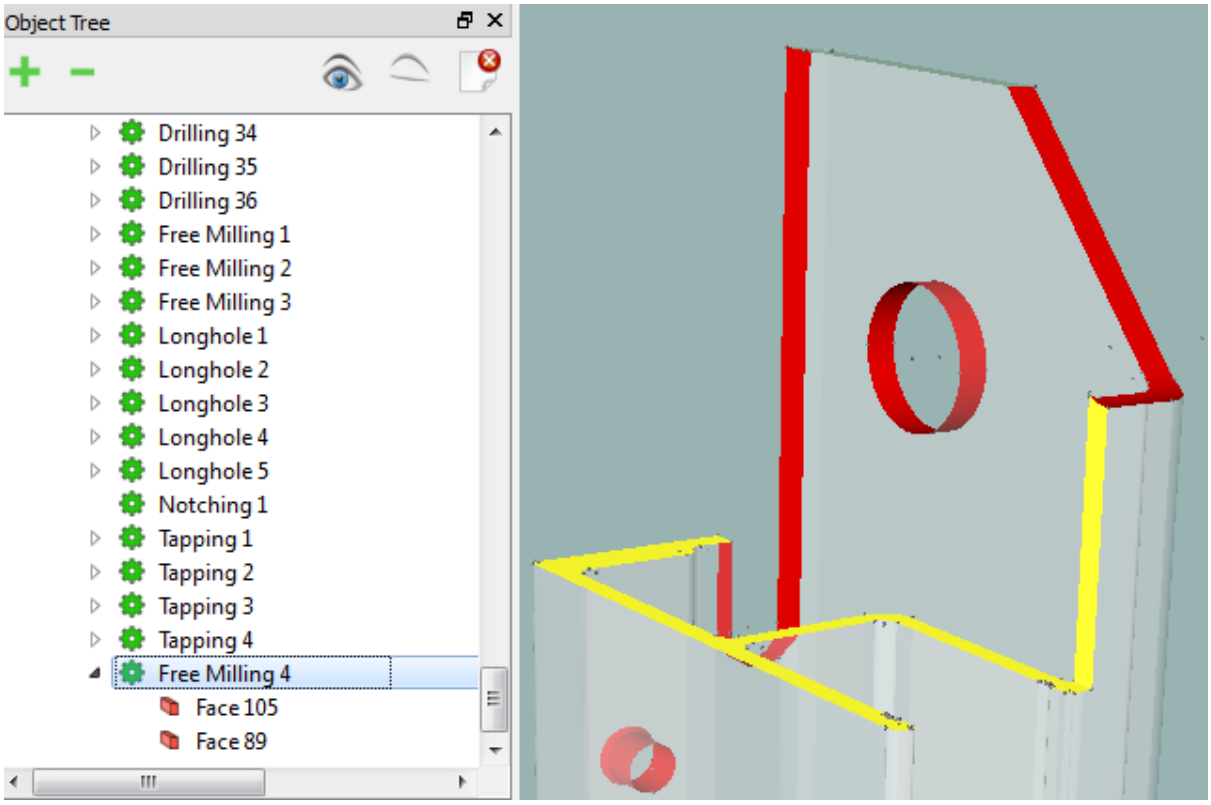
Select from menu:

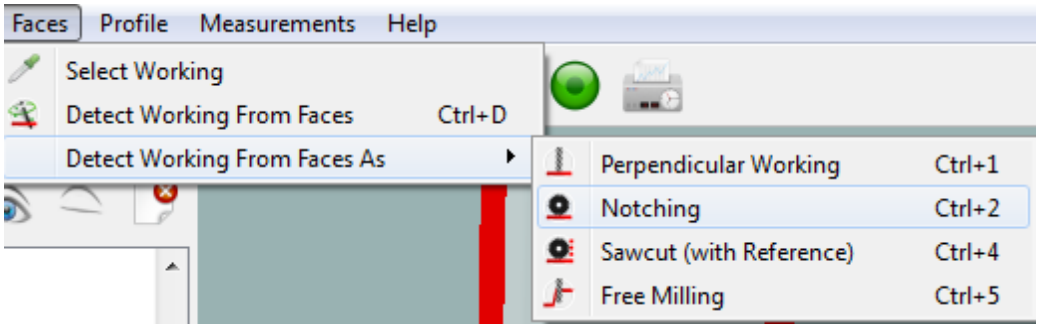


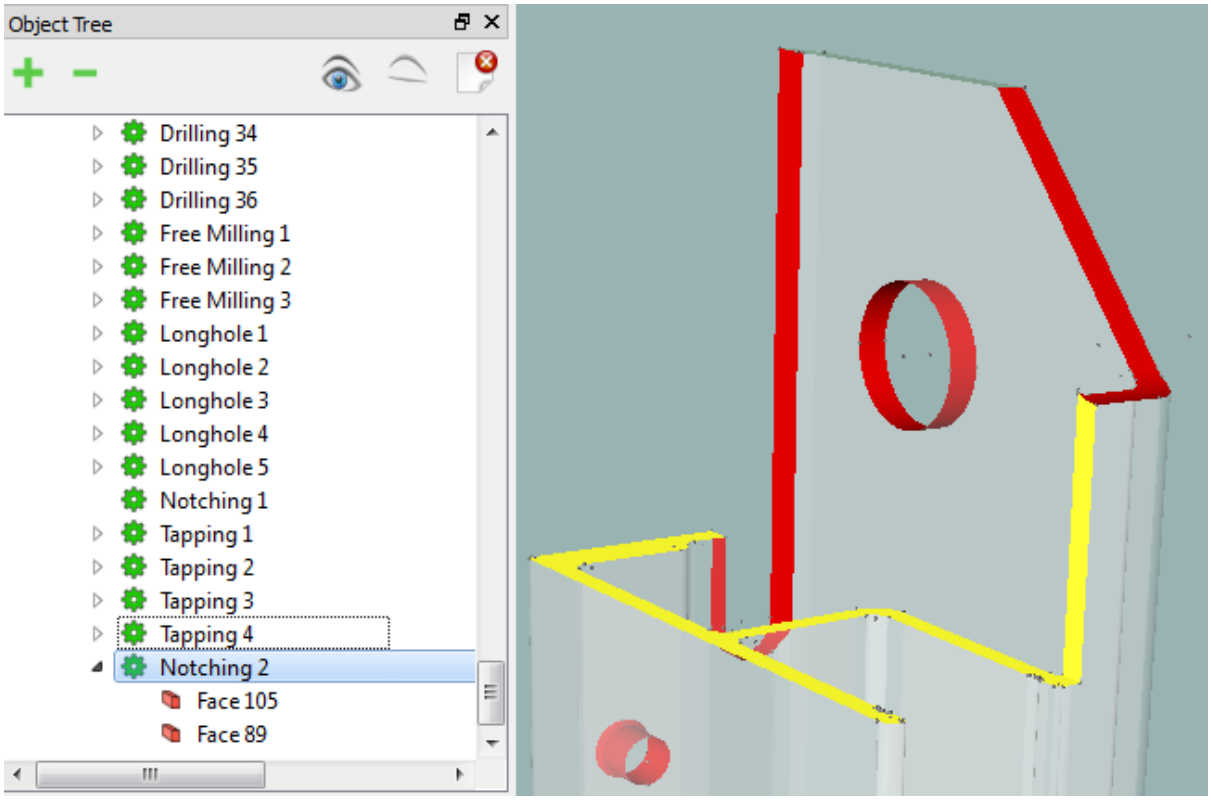
then:

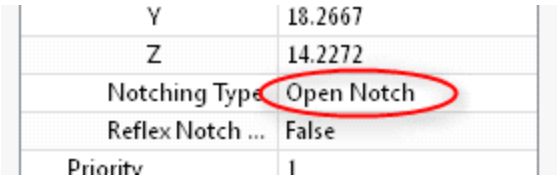
Detect Working As ... "Notching"

If the machine has got the possibility to handle notchings (5-axis-machine with a saw blade), you can select 2 faces (they does not have to be connected).









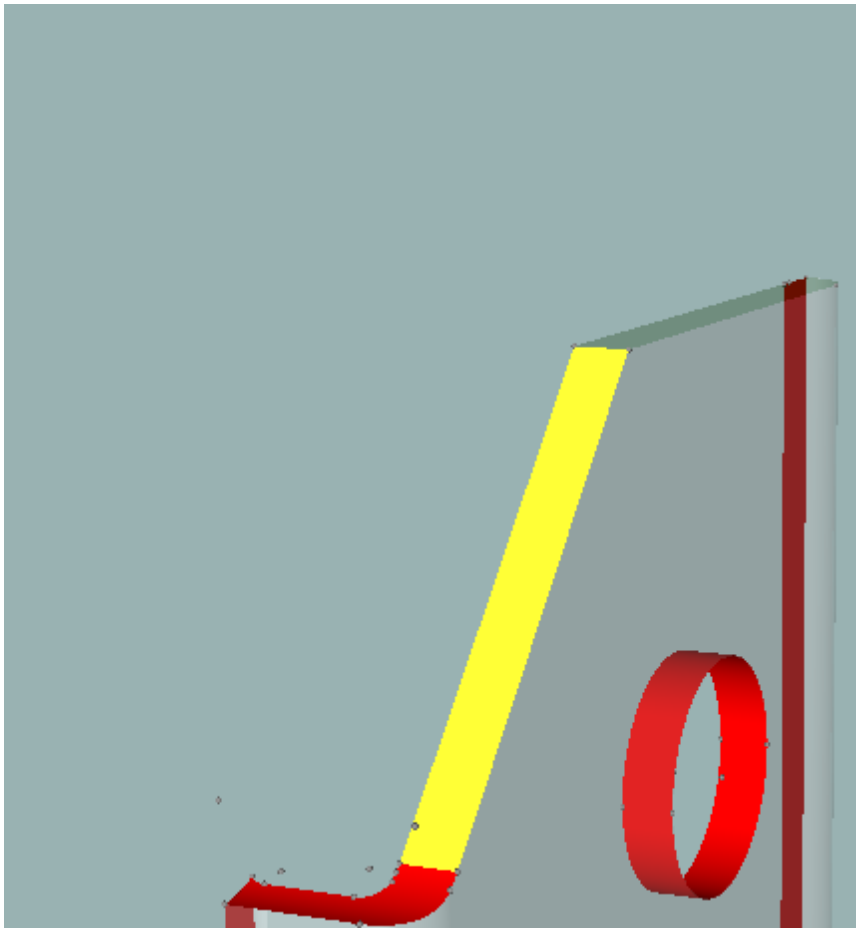
Out of these faces cadXtract generates a notching defined by the sawing line and the necessary angles, so you can use these data directly in camQuix to generate the sawing-cuts.

If you select only one face, cadXtract will generate a basic sawing cut (the sawing line will be calculated on the XY-plane at the bottom Z0).

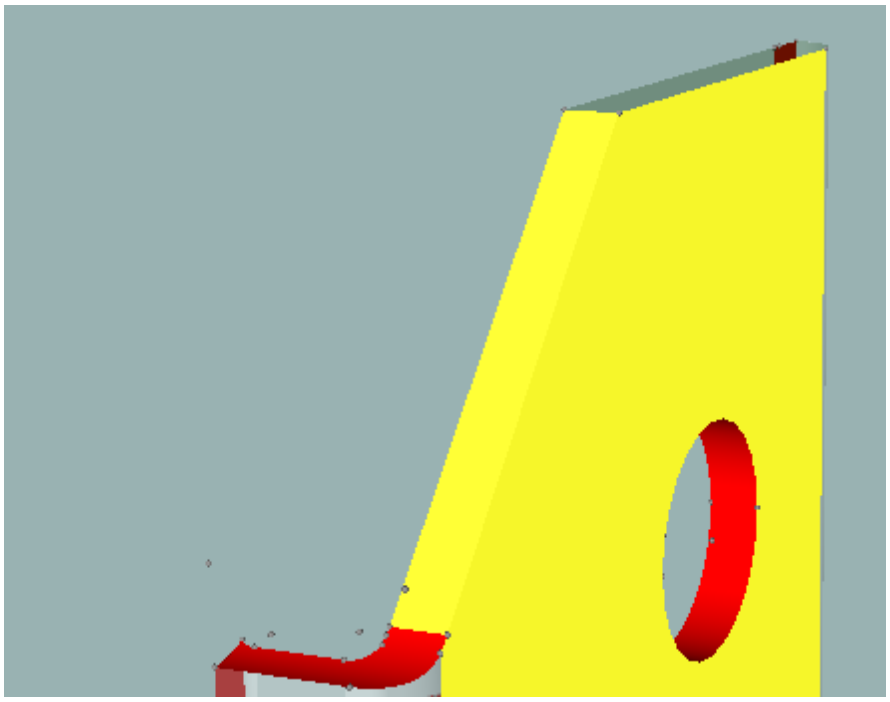
Detect Working as Sawcut

To Detect a working as sawcut, you have to do the following two steps:

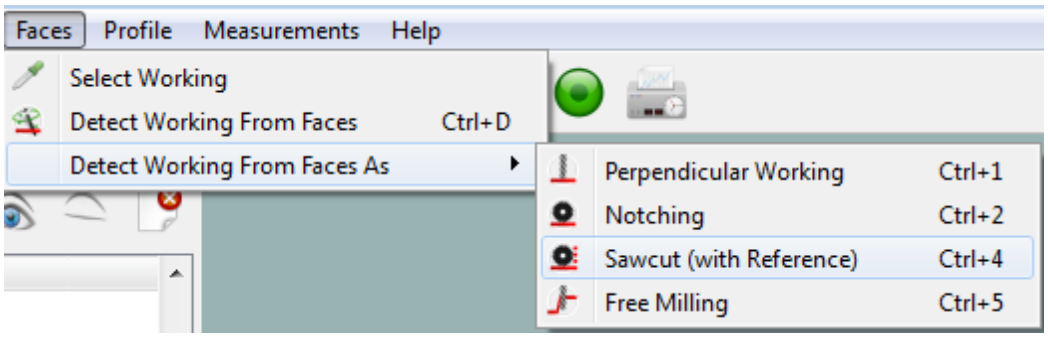
First mark the plane you want to define as a sawcut



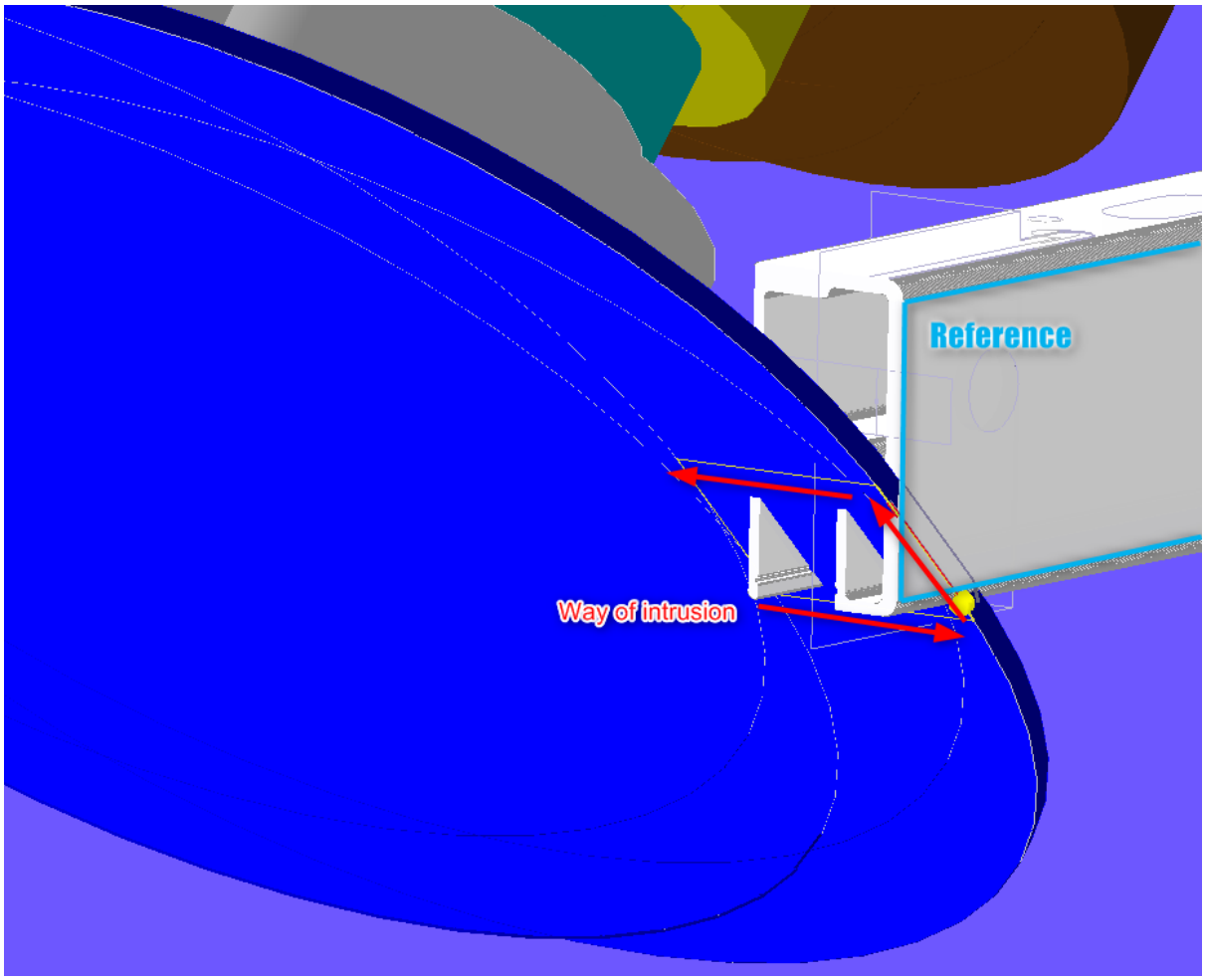
Then you can have two pick a second plane that works as reference for the cutting direction



After this, choose "Sawcut (with Reference)"

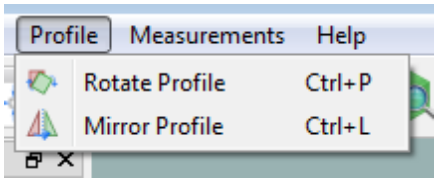


The second choosen plane as reference has the effect, that cadXtract will save the cutting direction. To explain this an easy way, the following picture from camQuix will show how it affects the way of sawing:



Part VIII

8 Profiles

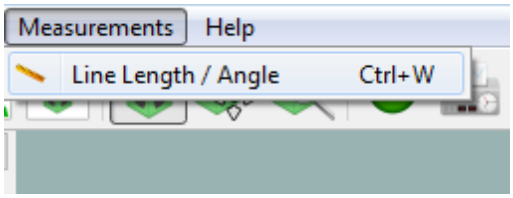


Rotate Profile: Rotates the profile so that its parallel to the machine bed. File must be reimported again.

Mirror Profile: Mirrors the Profile. File must be reimported again.

Part IX

9 Measurements

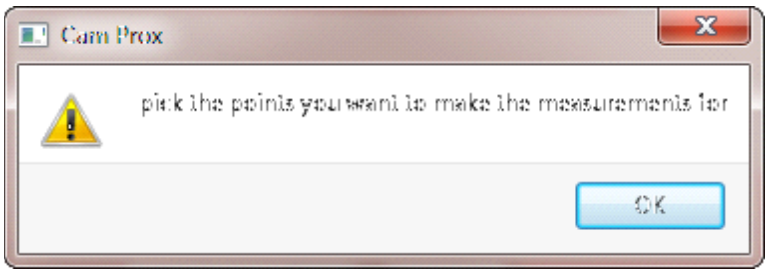


With these 2 functions, you can take out some measurements out of the model.

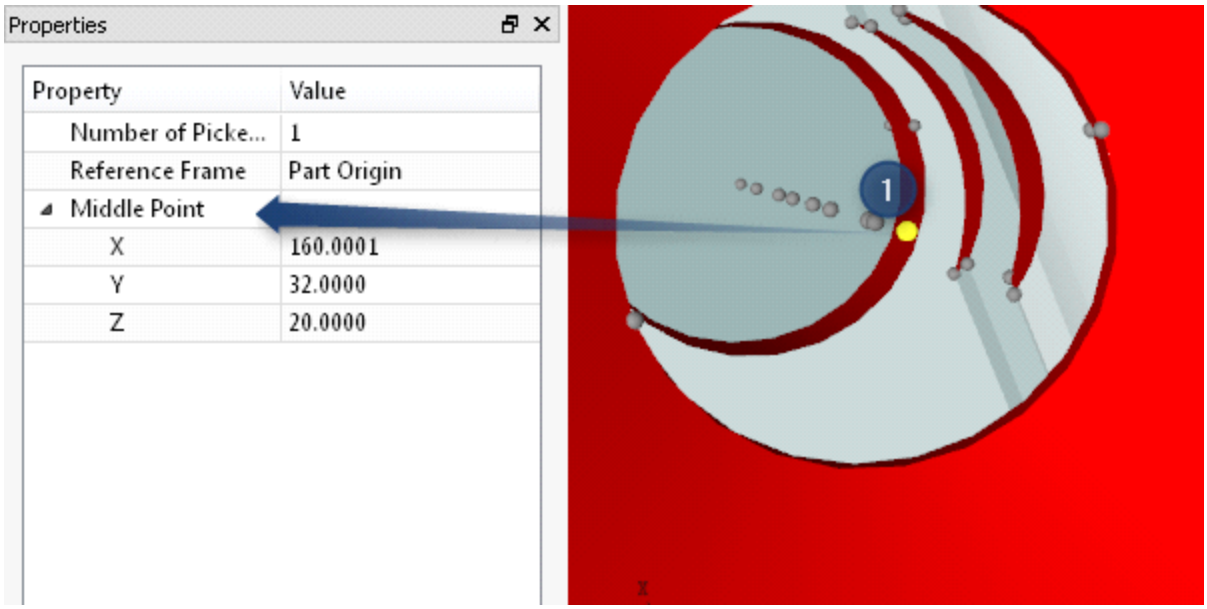
- Middle point calculates the data of the line from the 2 or more picked points

-With "Line Length/ Angle " you can take out the information about the distance from one point to another

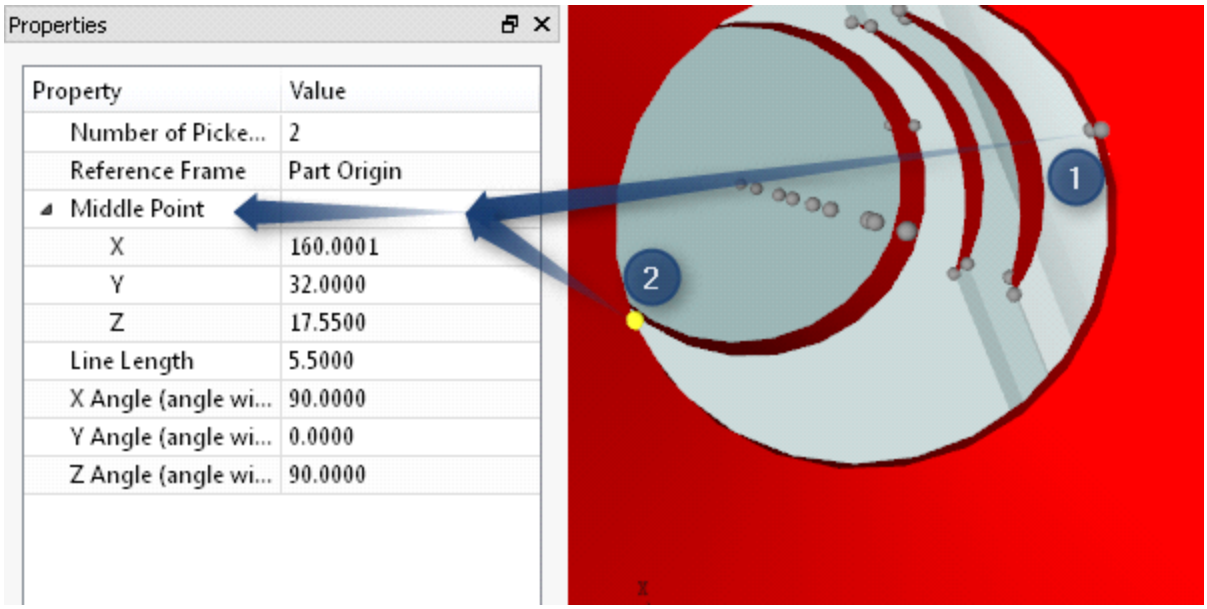
This function starts with this hint :



You pick the first point



and the second one :



So you can see (and copy) the distance and the middle point between those two points.

Part X

10 Symbols

Visible/Invisible/Delete

1 2 3

1 Make the actual face / operation / part visible

2 Hide the actual face / operation / part

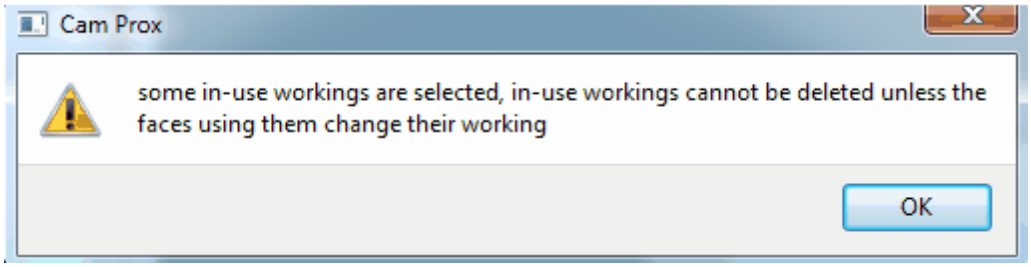
This can be used

- to get a **better view** (if more than one part has been imported), or

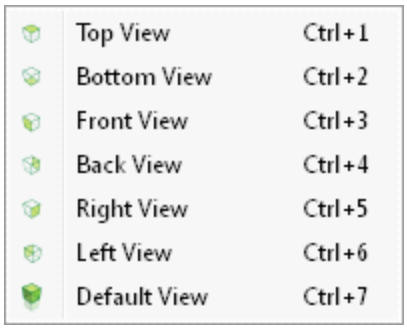
- to **suppress** this parts / operations from the export

3 Delete

-This is only possible, if the work has no assigned faces in the listview, else this message will appear :



Select the view direction



This direction refers to the coordinate-system of the construction and not of the actual part.

Set the camera target vertex

This means the picked point will become the center of the D-view

Fit All

This fits the actual assembly / actual part into the screen.

Zoom to object

This fits the actual view to the actual operation

Default View Behaviour

This option is used mostly, as you can move and scale the view by using the ALT-key and the mouse-wheel.
else you can switch to ...

Camera Pan

or…

Camera Zoom

Show Helper Points

Show Log

Description of the right Sidebar:

1	Create a macro from several workings
2	Check for macros again
3	Rotate the profile so that the selected side is parallel to the machine base. File must be reimported after rotation.
4	Mirror the profile. File must be reimportet after mirroring.
5	Select points to measure their distance, angle and middlepoint.
6	Invert the intrusion direction of the current working.

7	Rotate one-faced free millings by 90°
8	Combine the elected free millings to one working.
9	Detect the selected facings as workings.
10	Detect selected faces as perpendicular working.
11	Detect selected facings as notching.
12	Detect selected faces as sawcut. Pick a second plane as a reference. The intersection will be the cutline.
13	Detect selected facings as free milling.