MechanicalSample

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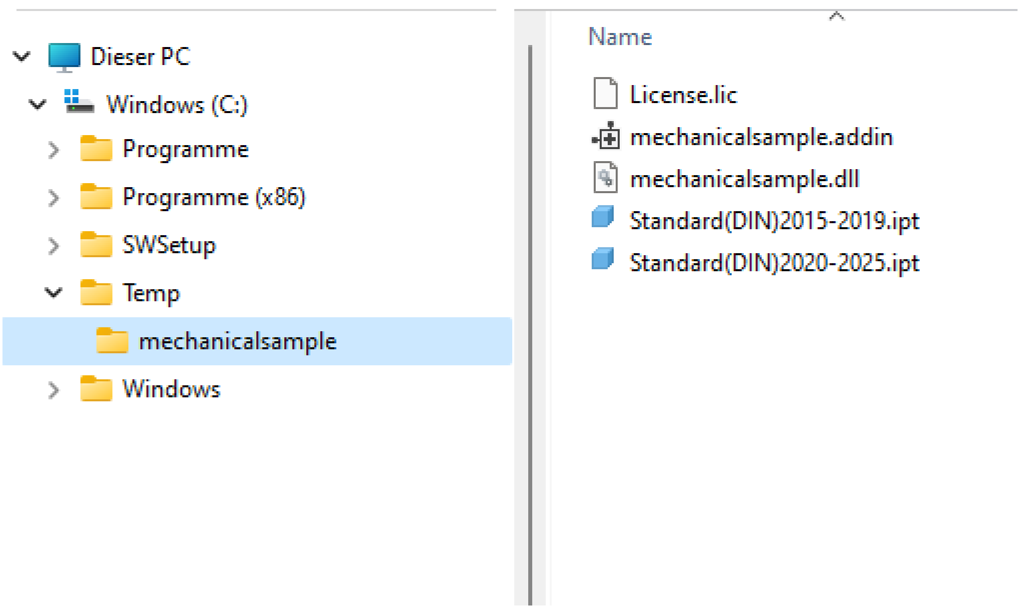
1.1 Info about MechanicalSample

MechanicalSample originates from the idea to facilitate the work with Inventor®.
Many users want a simpler handling in order to achieve a certain goal with as little effort as possible. Inventor® dialogs are often designed to be usable for many different users. For mechanical engineers, however, the work can be made a bit more specialized and thus more efficient. For this purpose MechanicalSample offers new commands in all areas. The most dialogs are in German and English depending on the Inventor® language version.

1.2 Install and set up

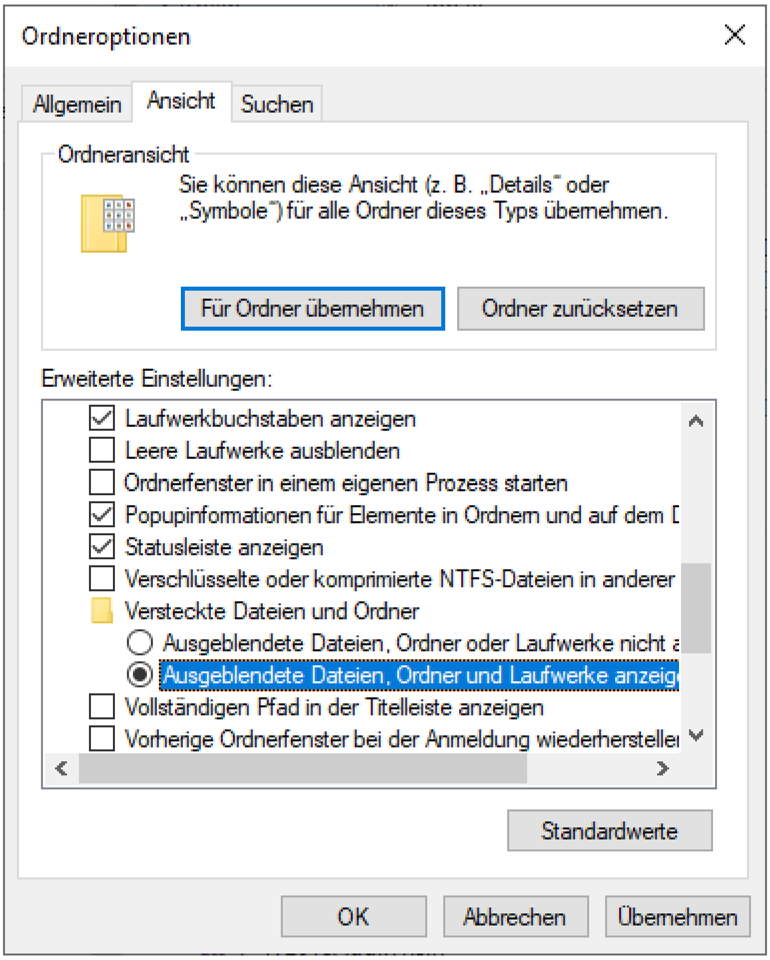
The MechanicalSample Addin can be downloaded at
https://mechanicalsample.com
The compressed file contains the MechanicalSample.dll and the MechanicalSample.addin as well as a standard Norm.ipt as delivered with Inventor® to enable a smooth first Inventor® start. Please note that the style definition of this Norm.ipt may differ from your company's version. After installation, a company-specific template.ipt can be set in the options, which contains your style definition. See the setting notes below.
MechanicalSample creates 3 new tabs. One tab each in Parts, Drawings and Assemblies.

Prepare:
Create a new folder: C:/Temp/MechanicalSample (be sure to spell it correctly!).
Download the zip file and unzip it.
Then place the unzipped files directly into the MechanicalSample folder.



In order to load MechanicalSample into Inventor®, the MechanicalSample.addin file must be copied into the Inventor® Addin folder.
The Inventor® Addin folder can usually be found, for example, in a local installation under:
C:/<your username>/Roaming/Autodesk/Inventor2024/Addins.
Or under: C:/ProgramData/Autodesk/Inventor2024/Addins .
You must have read and write permissions to one of these folders.
If the installation is not possible, ask the administrator of your organization if you are allowed to use MechanicalSample.

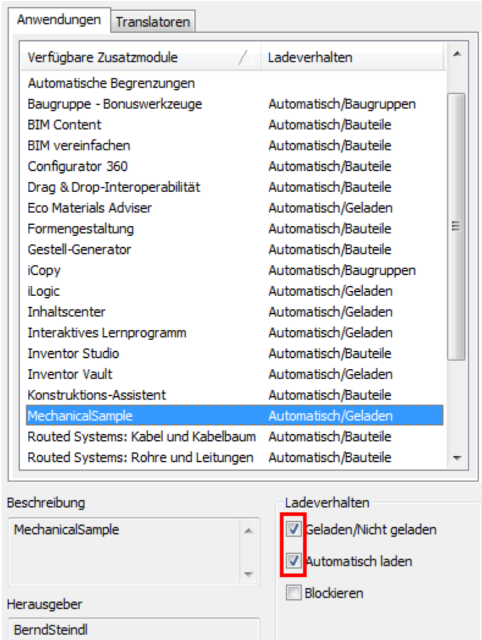
The Addin folder may not be visible. To show it you have to set the folder view "Show hidden files" in the file explorer.



If you do NOT want to install MechanicalSample on C:/, you must manually change the loading path stored in the MechanicalSample.addin file, according to your path where the MechanicalSample.dll file is located. To do this, simply rename the MechanicalSample.addin file to a .txt file and enter your path. Then rename the .txt file back to Mechanical-Sample.addin. Note that in MechanicalSample the drawing template reference is set to C:/. You can change this to your path in the MechanicalSample options available, after installation, in Inventor® so that MechanicalSample uses your own Template.ipt.
FOR EXAMPLE: C:/VAULT_Work/PDM_Vault/Templates/Inventor/Templates/your_work.ipt

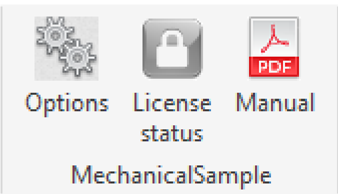
Once the preparations are complete, start Inventor®.

At the first start with MechanicalSample you have to unblock and set the loading behavior. (Inventor® button: Tools->Additional modules->MechanicalSample).



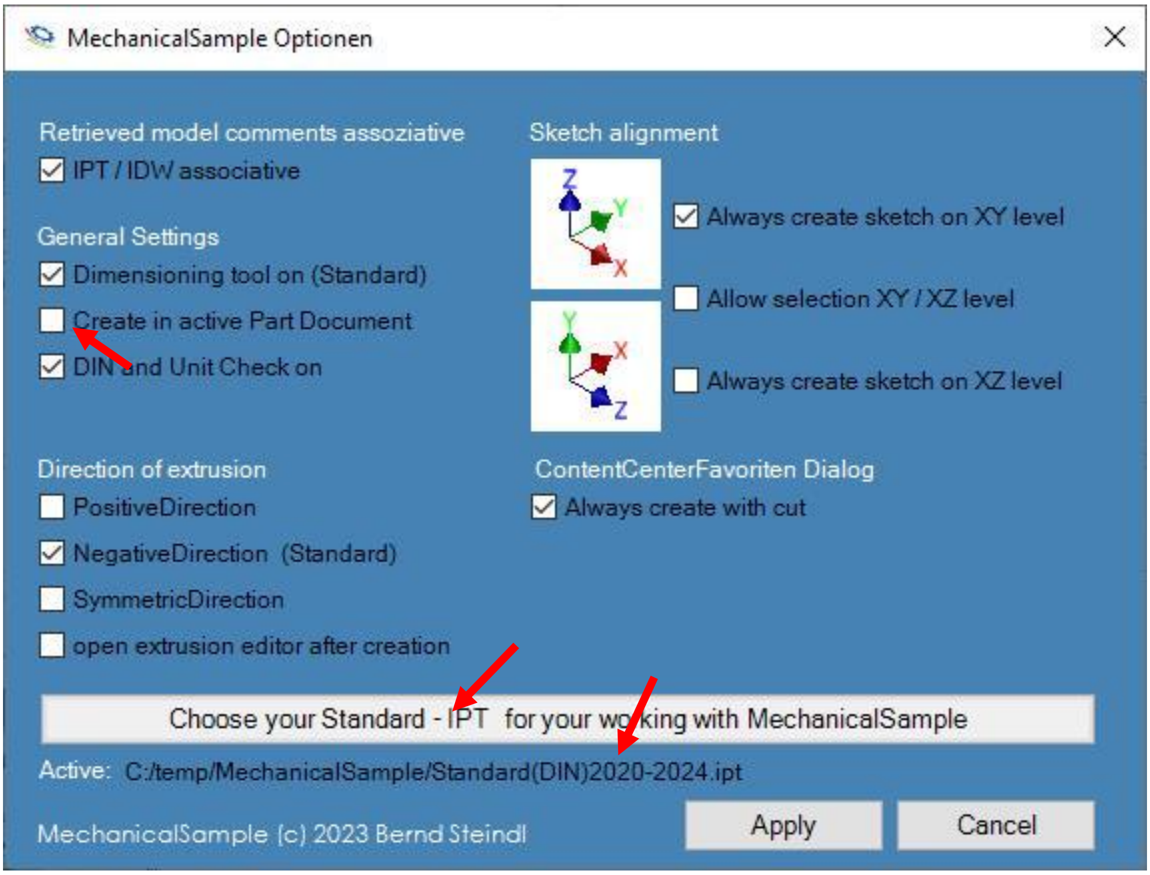
Afterwards you have to accept the license conditions in the following dialog to be able to use MechanicalSample.

Once Inventor® has been started and the "Options" and "Manual" buttons appear in the "First Steps" tab, MechanicalSample has been installed and work can begin.

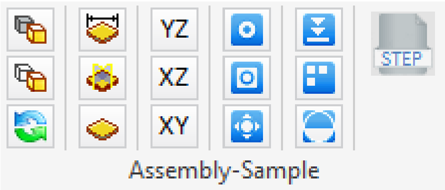


To be able to use MechanicalSample in your company, there is the possibility to define your working .ipt under the options. If new parts are created with the included volume tools for cuboids and rotations, this file is used as a template file. However, you can also set MechanicalSample to create parts in the active file that you have opened.
Furthermore, planar sketching can be changed when creating cuboids on the X/Y plane (standard Inventor®) or on the X/Z plane for cuboids. Additionally, the default extrusion direction can be set for cuboids.
These are the most important settings you should make.

Options Dialog



1.3 Commands for IAM's



Cube Icon "visible"

Makes all "not visible" set components visible again with one click

Cube Icon " suppressed "

Makes all "suppressed" components visible again with one click

Swap icon

Replaces components inserted in an assembly with an existing component. The dependencies of modified parts, which originate from the part to be exchanged, are also taken over.
The selection is made as follows:
1. select the component installed in the IAM. 2. select the component inserted in the IAM.
2. select the part inserted in the IAM which is to be exchanged with the 1st part.

Sketches

The sketch icon can be used to switch on the sketches of individual components in an IAM.
After clicking on the symbol, select the corresponding component.
If a component within the IAM is in "edit mode", this component is automatically selected to switch on the sketches for editing.

Switch on Workplanes

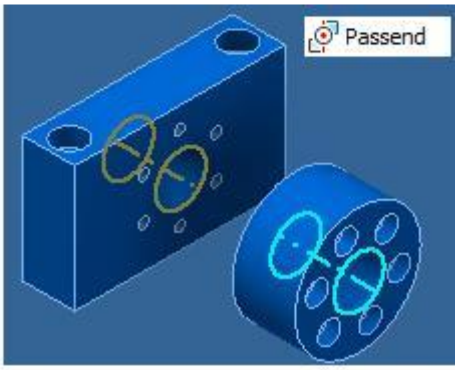
The layers of individual components in an IAM can be switched on using the layers icon. After clicking on the symbol, select the corresponding component.
If a component within the IAM is in "edit mode", this component is automatically selected to switch on the layers for editing.

Clean icon

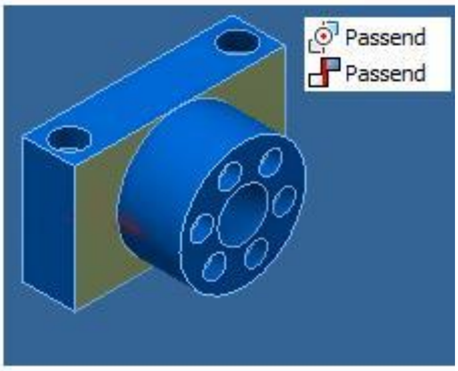
The cleanup icon offers several options in assembly's.
The 1st click turns off the sketches in the IAM in all parts.
The 2nd click turns off the layers in the IAM of all parts.
For assemblies with many parts, this can cause delays until everything is processed.
If a component within the IAM is in "edit mode", this component is automatically selected and the cleanup only applies to this component.

Assemble parts

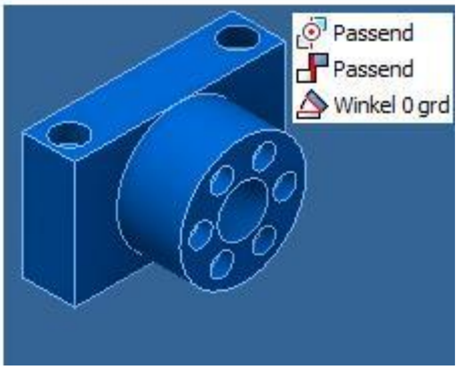
With these 3 buttons it is possible to create axis dependency, area dependency and angle dependency in one single step. The prerequisites are holes with which the components are to be connected to each other. This is true in 90% of the cases. By selecting edges instead of surfaces, it is not necessary to rotate the parts to a different position because edges of holes can be selected even behind the parts.



Axis dependence only



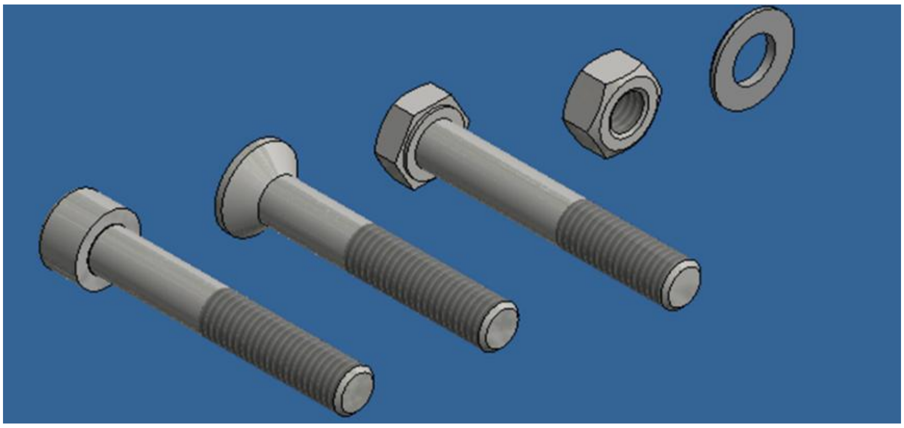
Axes and area dependence only



Axis, area and angle dependence

Favorite icon for components from the content center

MechanicalSample offers the possibility to efficiently display bolts or nuts from the Content Center as favorites in a dialog and to automatically install them in IAMs with a 2-click method with the required size and length. The parts are thereby axes, surfaces and angle-dependent installed.
In order to make the tool usable for different libraries in different countries, a desired component from a category of the content center must first be inserted in an IAM in any size, in order to make it "known" to the favorites tool with the help of a "scan process".Ideally similar components as shown in the picture.

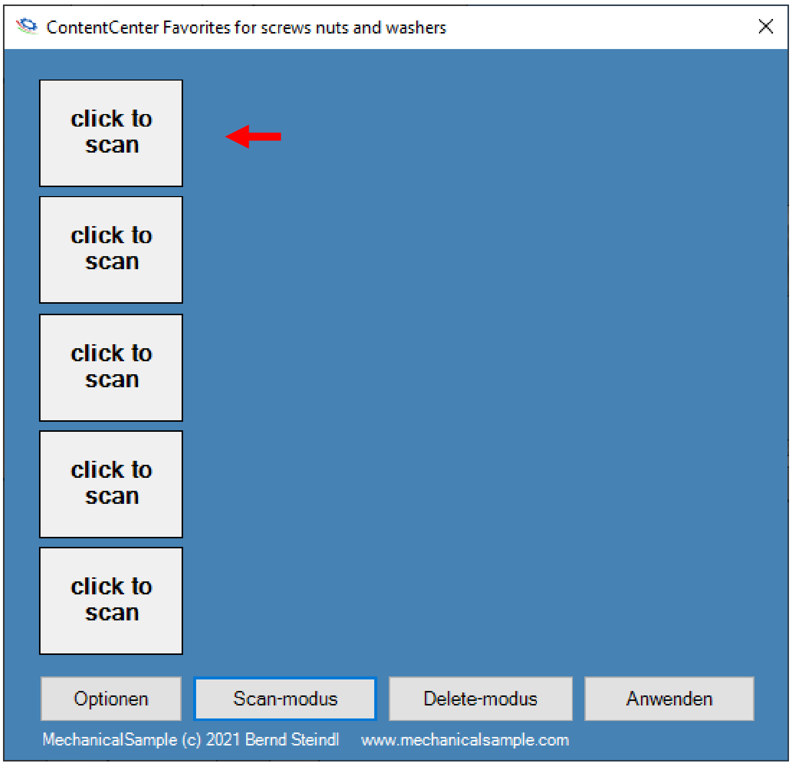


After the "scan process", all sizes of a category of CCParts are recorded and available. During the subsequent assembly in an IAM, the CCPart is taken directly from the content center. This means that the CCParts always remain up to date, even if changes are made in the content center.

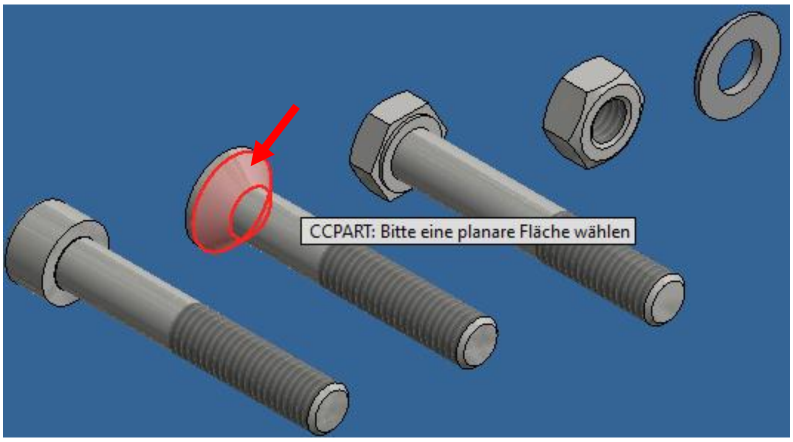
The size and length of the screws are determined automatically, regardless of whether the screw countersink or the screw hole was clicked.
It is also not necessary to find the so-called "first hole in a round or rectangular arrangement.
The tool is also able to detect all the same holes in a plane and fill them with a CCPart, even if there is NO arrangement for it.

Setting up and using the Favorites tool:
A maximum of 5 favorite parts can be scanned.
As a first step, the data of the CCPart must be collected or scanned with the help of the favorite tool. In particular, the insertion area must be defined. The insertion surface must have a flat circular surface or a conical circular surface. Afterwards, the CCPart can be installed directly from the Favorites dialog. Arrangements are taken into account in the same way as identical holes in the selected plane.

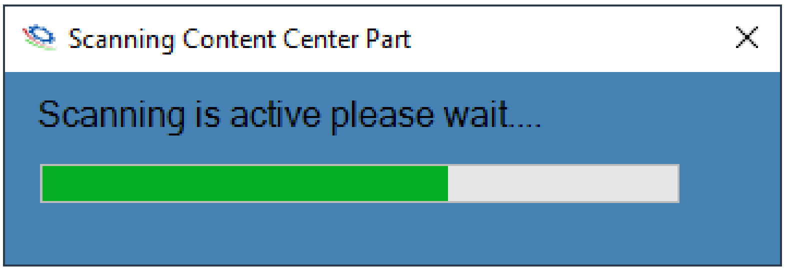
How it works:
Open the Favorites tool, activate scan mode and click on a button labeled "Click to scan".



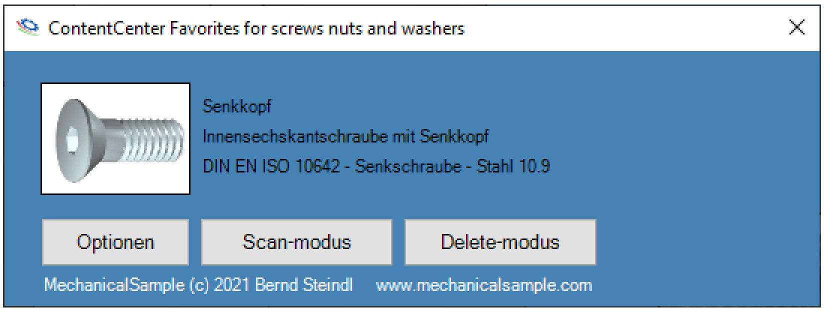
After clicking on one of the buttons, the desired insertion area must be selected on the CCPart. Then the scanning process starts. Depending on the computer performance, the scanning process can be longer or shorter.



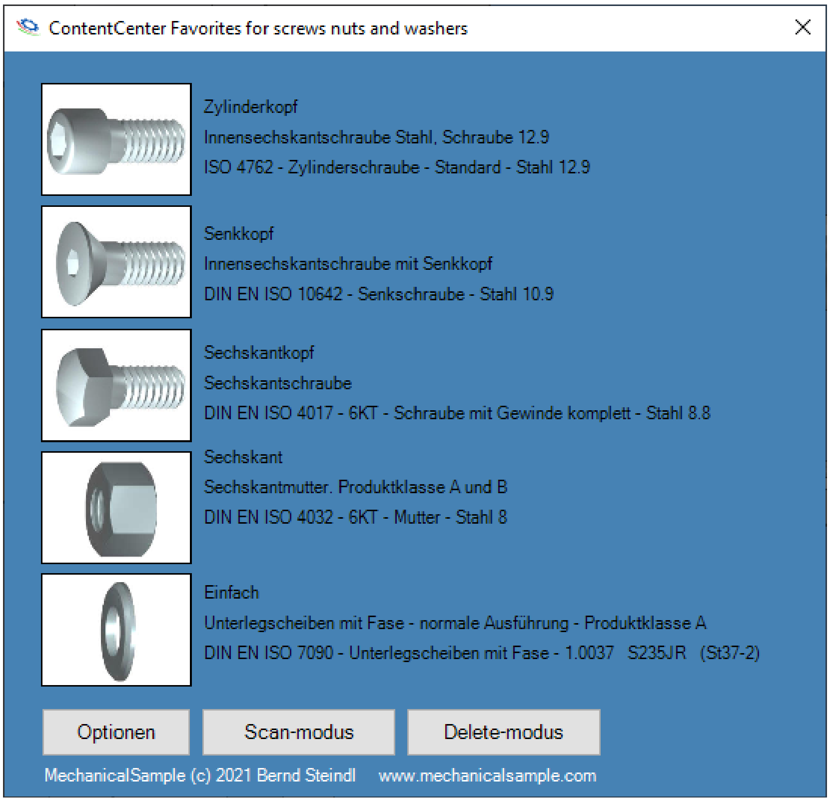
Scanning process



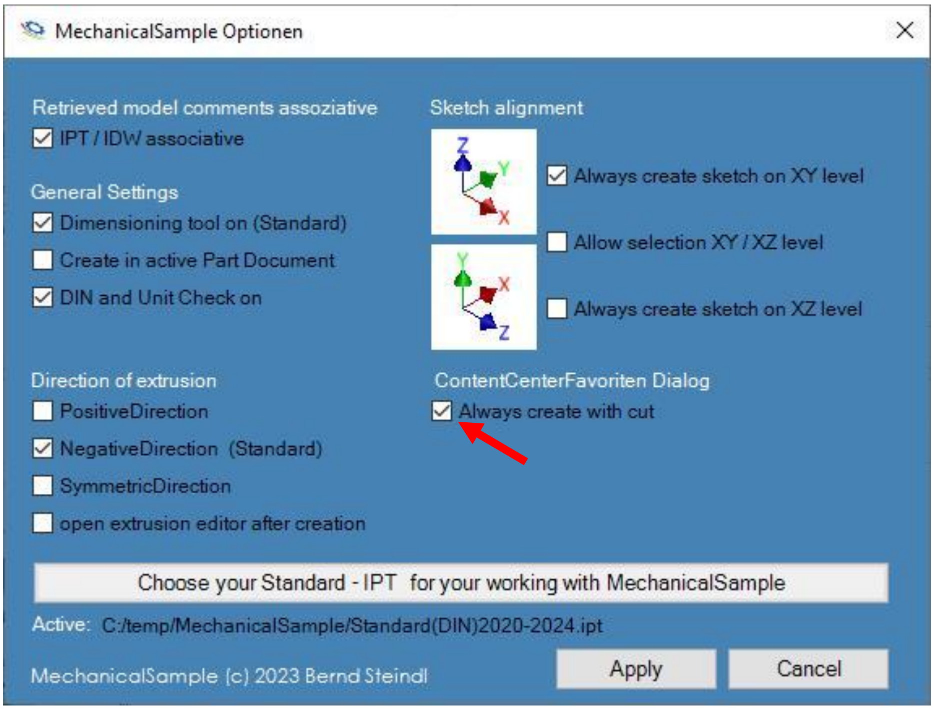
By clicking on Apply after the "Scan process" the configuration is completed.
By reopening the Favorites dialog, the CCPart is immediately available for installation in IAMs. The corresponding button is displayed.



Additional components can be added by switching on the "Scan mode" and clicking on a free button. The Delete mode can be used to remove individual components from the Favorites tool.

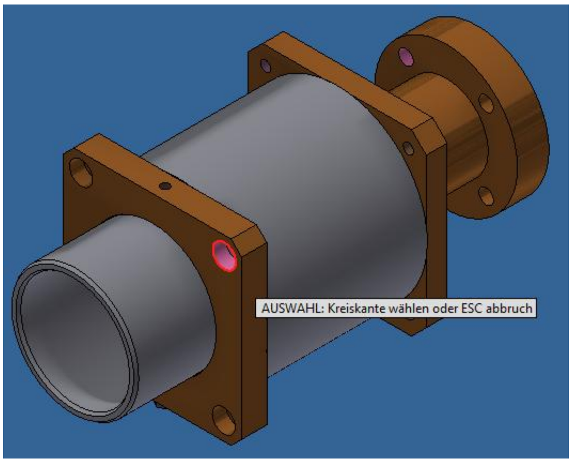


By default, after selecting a hole, a section through the hole is created to show the shoring of the CCPart. However, the command can also be used WITHOUT displaying the part in a section. In this case the length is determined by selecting an outer edge. This can reduce the shoring process to 2 clicks, drilling and outer edge. With the " Cut on Edge" button the result can be displayed later if desired.
Switching off the cut display is set by deselecting the "Always create with cut" check box in the options.

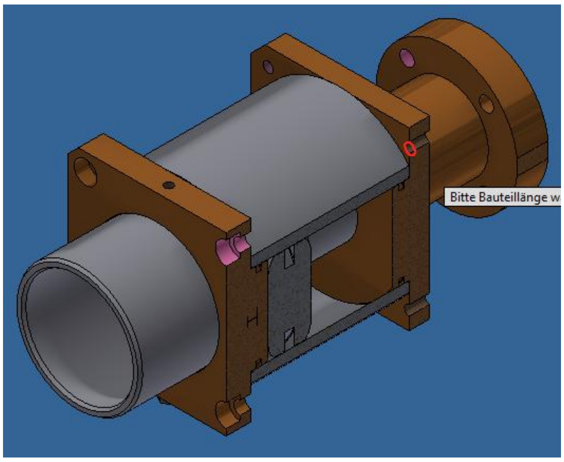


Application examples with the option "Always create with cut"

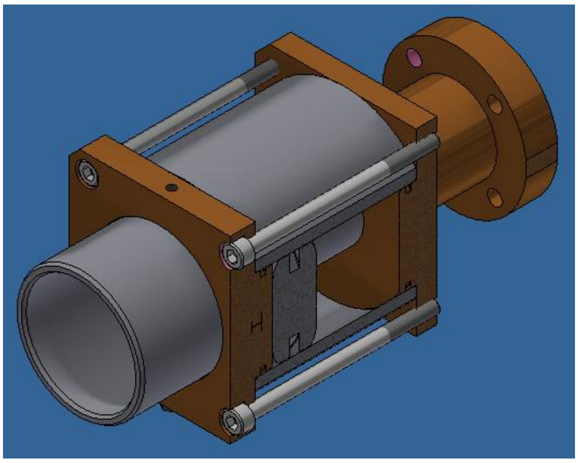
Select counterbore



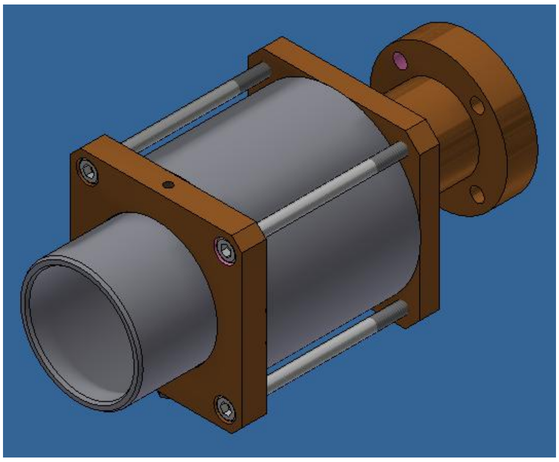
Select thread start



Screw is inserted automatically

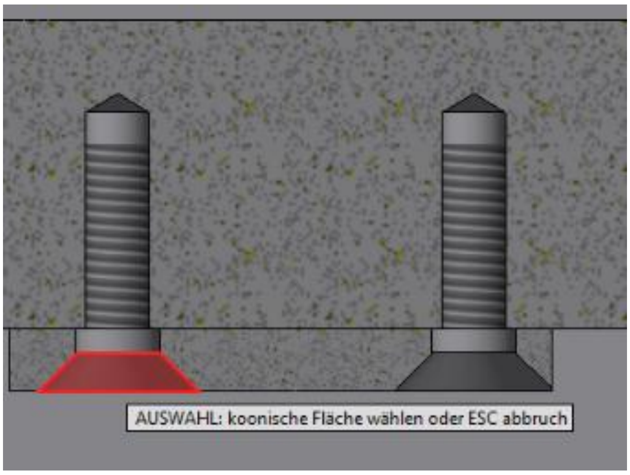


Confirm "Apply" dialog

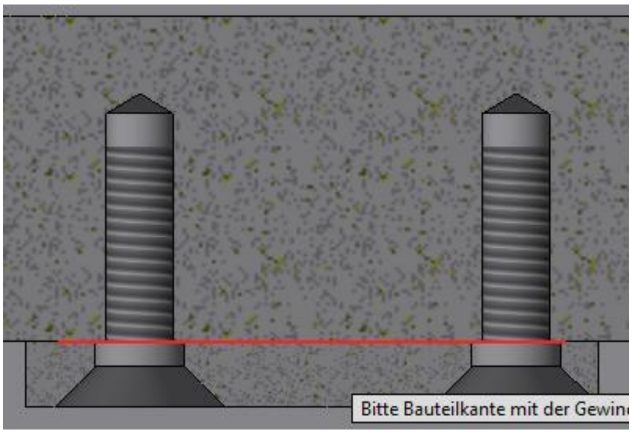


Application Example with countersunk screws:

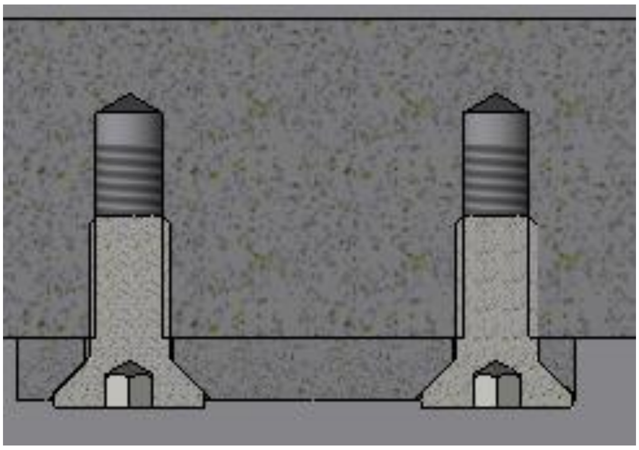
Select conical surface



Select edge of the thread start

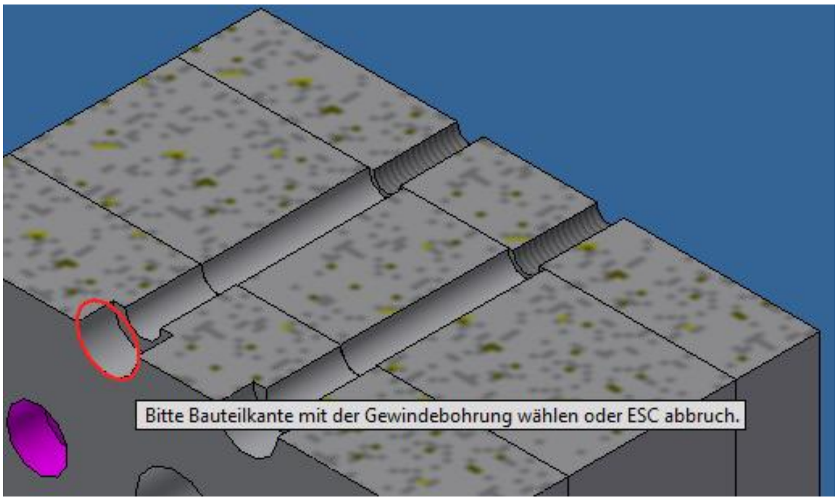


Screws are inserted

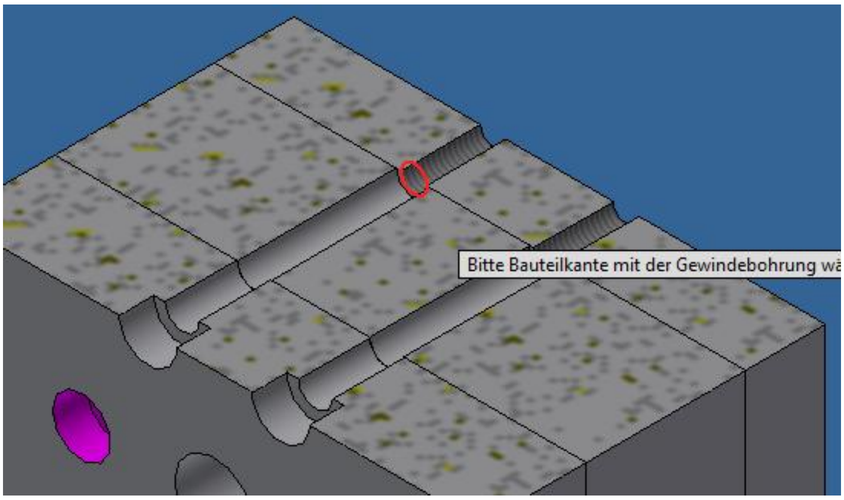


Application example with intermediate plates:

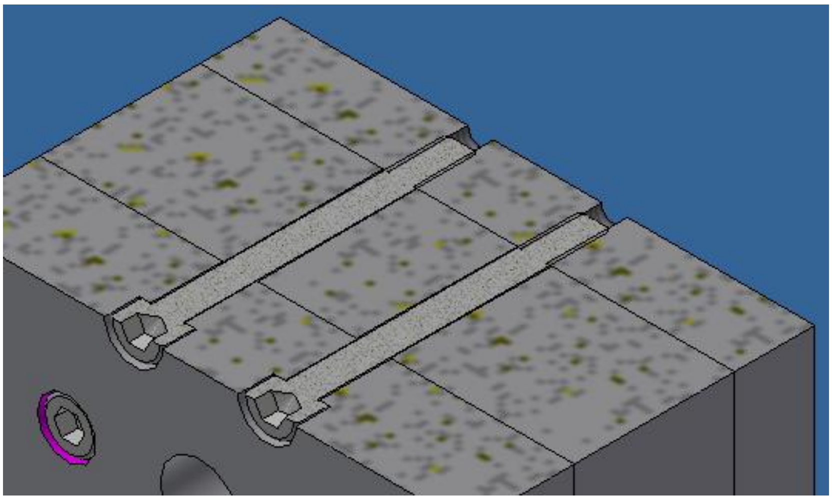
Select circular area (At the sink or in the sink)



Select length of shoring:

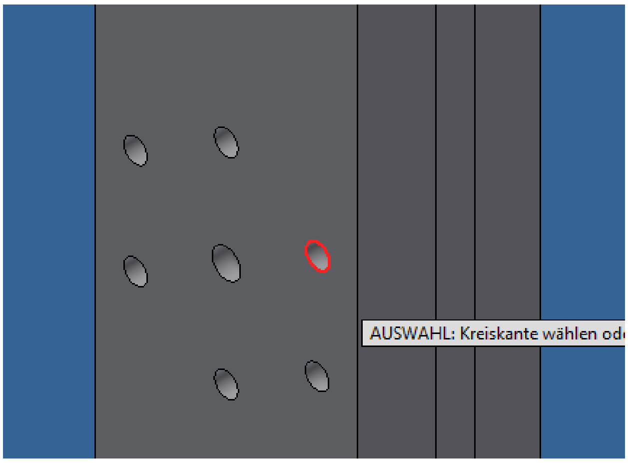


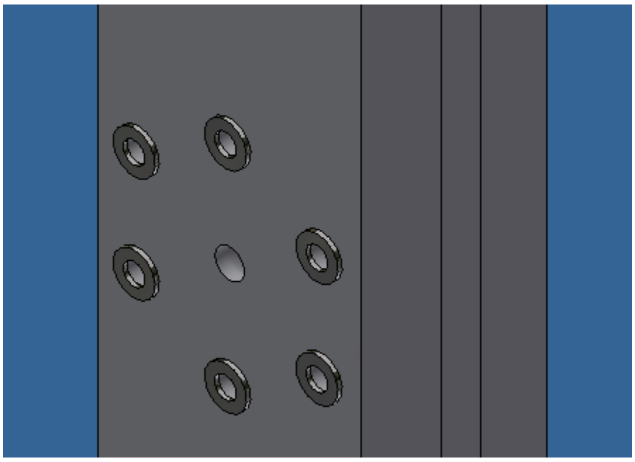
CCPart's are inserted (here the shoring in a round arrangement)



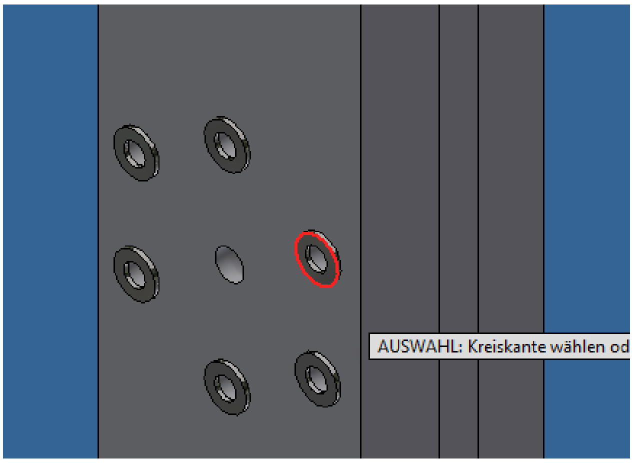
Application example with washers

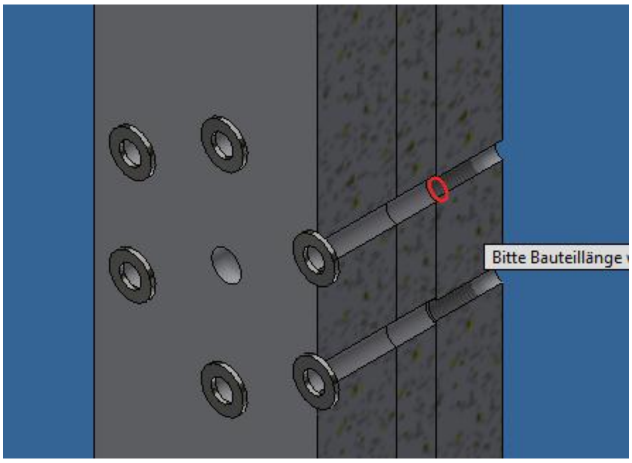
1.insert washers in round arrangement



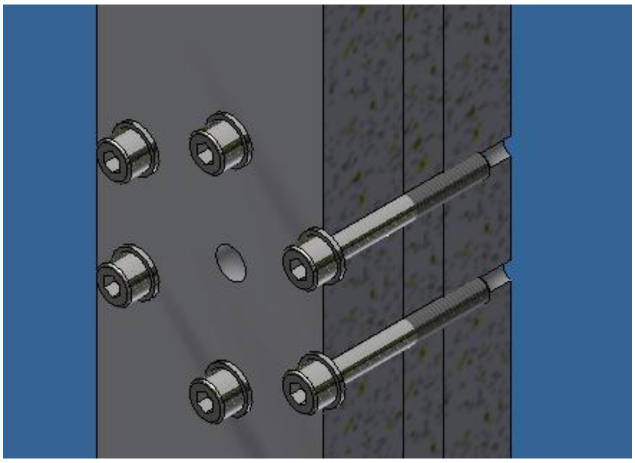


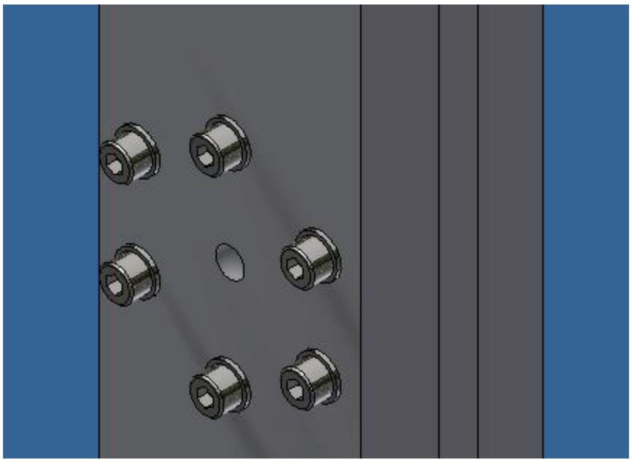
2.insert screw by clicking on the washer



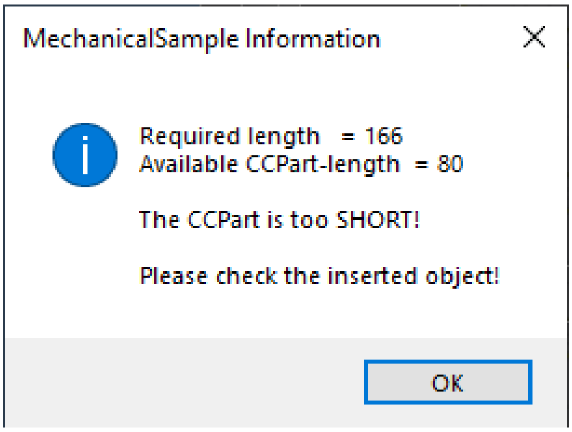


3. complete the action by clicking on "Apply



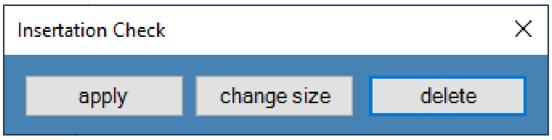


The Favorites tool examines the available lengths of a CCPart and displays a warning message if the length does not fit the shoring.



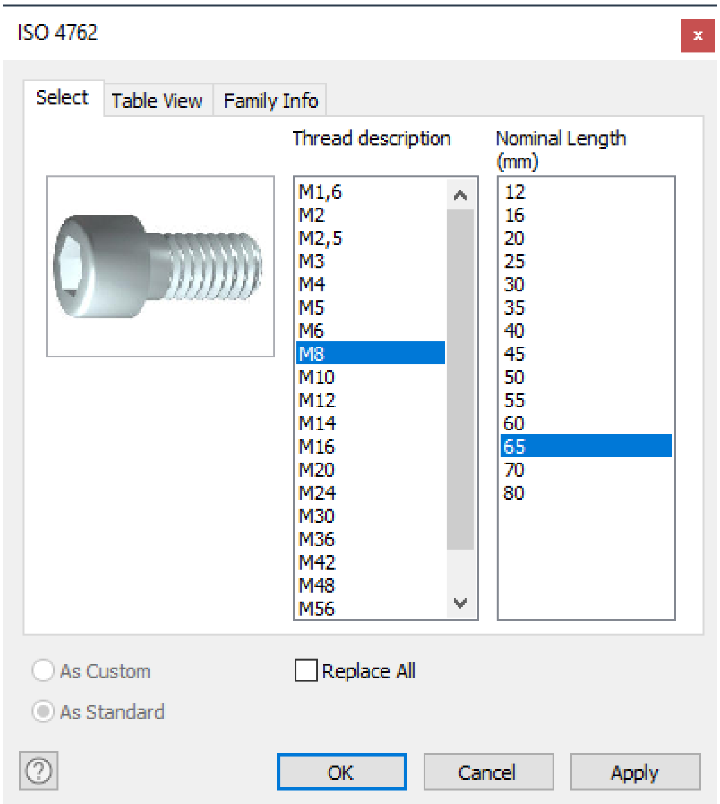
The inserted screws can be resized, if necessary, with the Inventor® command "Resize... " command to resize the screws

After the insertion operation the dialog "Insertation Check" appears.
The dialog offers 3 options:

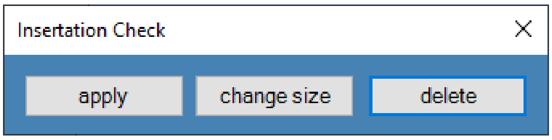


"Apply" terminates the command with the user input.

"Resize" calls the Inventor® dialog "Resize..." to make entries.

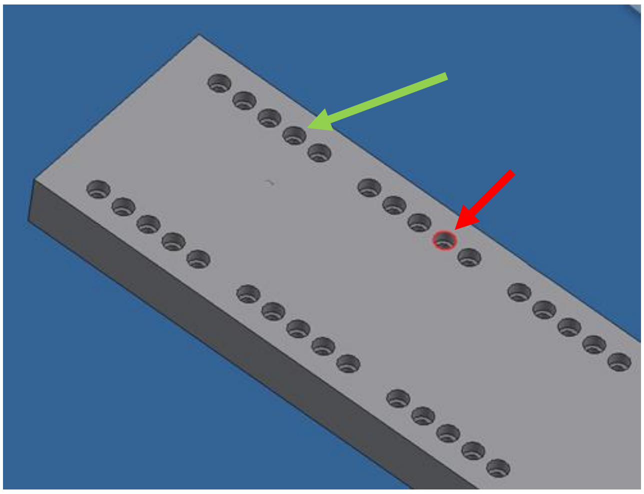


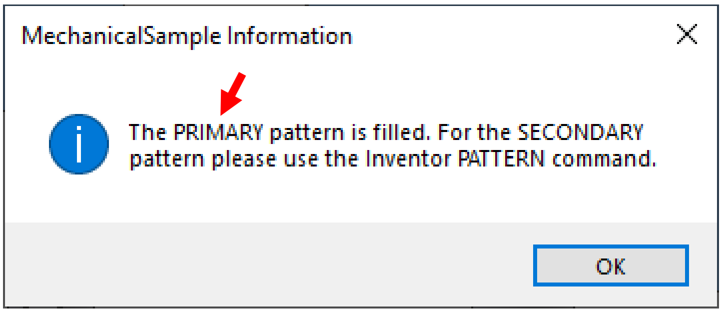
after the entered size changes the CCParts are changed accordingly and the dialog "Insertation Check" appears again.

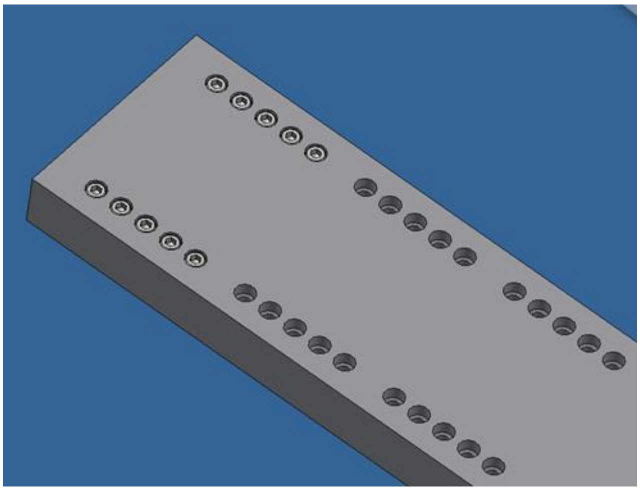


"Delete" terminates the command and deletes all user input.

The Favorites tool fills primary pattern only. Secondary patterns can be easily added after filling the primary pattern using the "Inventor Pattern" command. When a hole is selected in a secondary pattern , a dialog opens with a note that only the primary pattern will be filled.

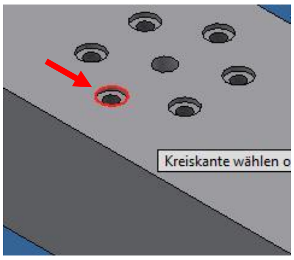


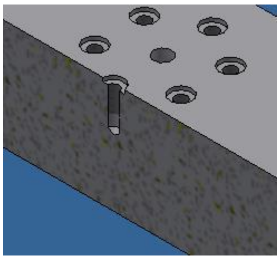




Cut on Edge Symbol

With "cut on Edge", cuts can be created along a component with a circular edge.
This is useful to check what is in a hole or if, for example, the inserted screw was created correctly. Simply select a circular edge and the cut is generated. The tool recognizes from which direction the user is viewing the part and creates the cut according to the viewing direction.



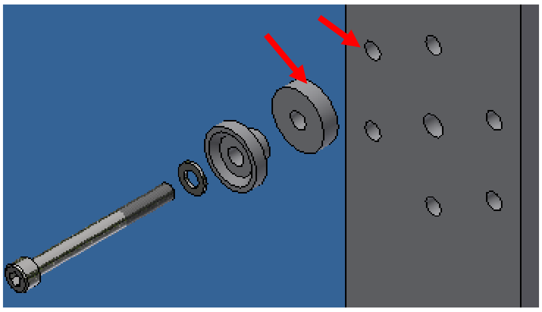


Multi-Insert-Symbol

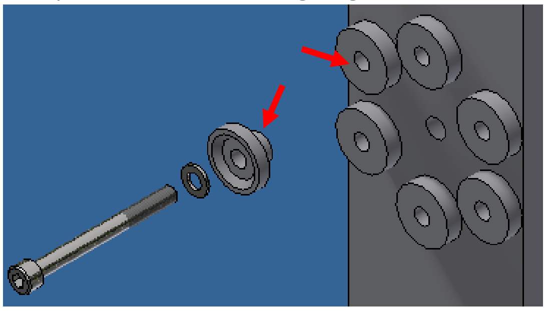
With this button, single IPT's, several assembled IPT's, or inserted IAM's consisting of several IPT's can be assembled in hole arrangements or in the same holes with 2 clicks.

Application example:

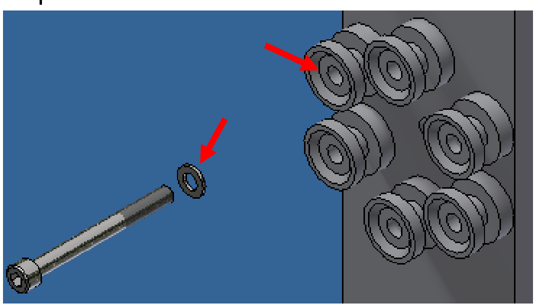
Step1: Select edge of 1st slice



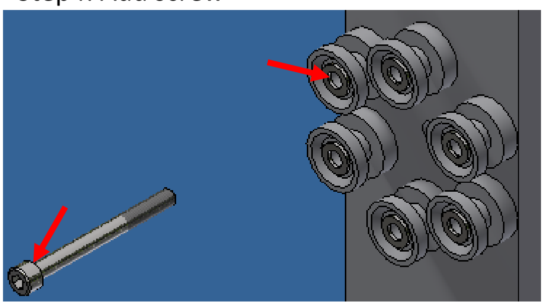
Step2: Select connecting edge of 2nd slice



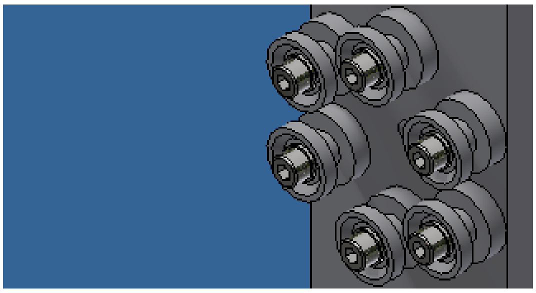
Step3: Connect washer



Step4: Add screw



Step5: Action is completed

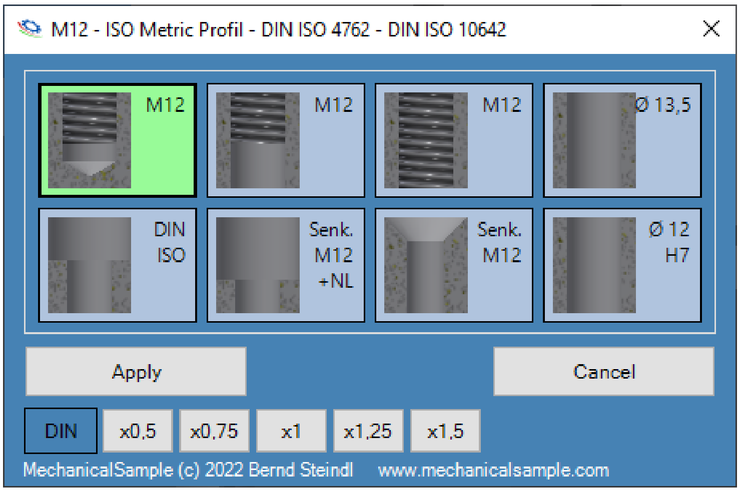


1.4 Commands for IPT's

The drilling tool

........................to..............................

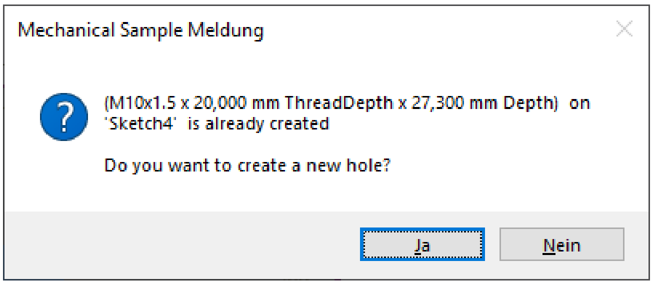

Clicking this button opens an Inventor® button menu to display a hole dialog from M3 to M24. Threads are created according to the ISO Metric profile and dimensions as defined in the Inventor® Thread.xls file. Countersinks are created for socket head cap screws DIN 912 / DIN EN ISO 4762 and countersunk screws with hexagon socket DIN 7991 / DIN EN ISO 10642.



z.B ->M12

In the dialog you will find further selection options that refer to this preselection.
Thread blind hole, thread passage with thread depth, through thread, screw hole, counterbores for cap screws, counterbores for countersunk screws, and pin hole. When a thread field is selected, a selection of available thread pitches also appears. After clicking on one of the selection fields, a first hole is created on a sketch point to be selected and the Inventor® drilling dialog opens for further selection of sketch points for drilling.

If an identical hole is found on the same sketch, MechanicalSample asks the user if the same hole should be reused or if a new hole should be created, independent from the found hole, e.g. to set the thread depth individually independent. This approach allows to place all sketch points for holes on a single sketch. The drilling tool can also be used in IAM's when a part is in edit mode.



If the creation of a new hole is denied, the existing hole is activated and the hole dialog opens for further selection of sketch points.

Edge break 45°

Clicking on this button opens an button menu to create a 45° chamfer in a pre-selection of parameters. The command can also be used in IAMs when a part is in edit mode.

Edge break 30°

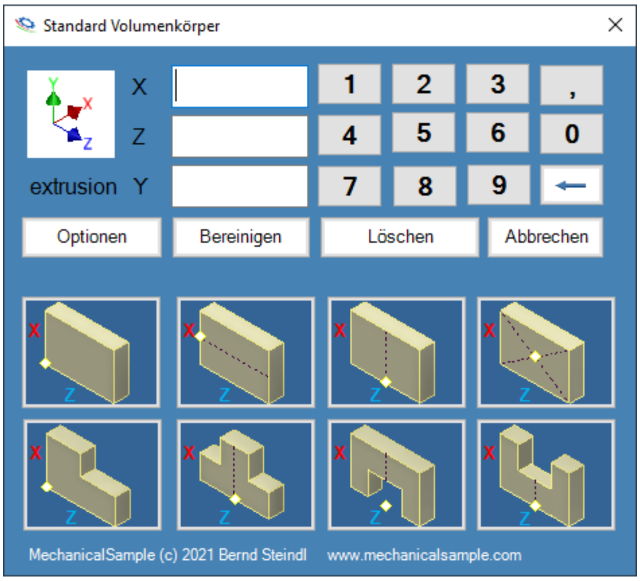
Clicking this button opens an button menu to create a 30° chamfer in a pre-selection of parameters. The command can also be used in IAM's when a part is in edit mode.

Radii

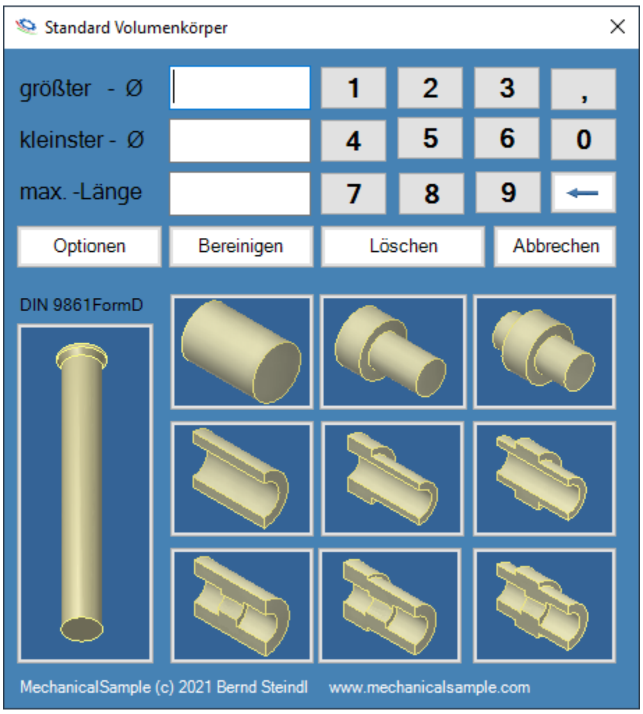
Clicking this button opens an button menu to create a radius in a pre-selection of parameters. The command can also be used in IAM's when a part is in edit mode.

Dialogs for standard solids.
With these two tools, parts can be created in a simple way with a completely defined sketch. For this only 3 parameters are necessary By entering X,Y,Z
and the subsequent click on a component symbol with the approximately desired contour, the volume is generated fully automatically. After the volume has been created, the component can be easily edited in sketch mode to the desired dimensions.

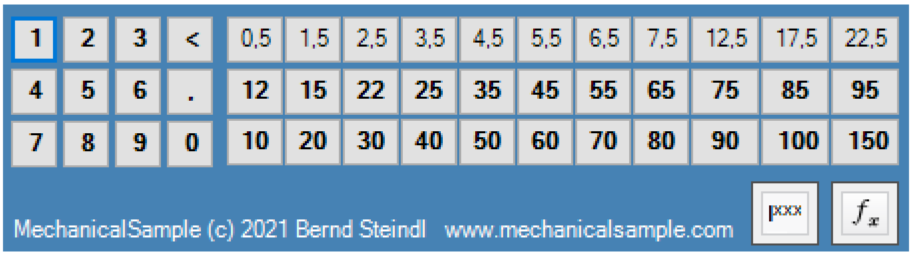
Dialog standard cuboid solid



Dialog Standard Cylinder Volumetric Body

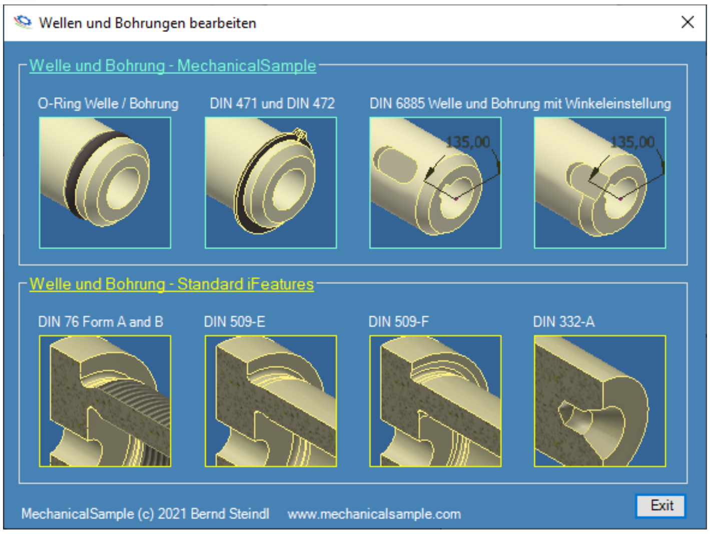


The dimensioning aid:



If a sketch dimension is changed, a dimensioning aid is switched on, which takes over the control of the dimension input. Here, individual dimension entries can be made with the mouse or default parameters can be selected directly. The dimensioning help can also be switched off in the options under General settings->Dimensioning tool, if this is not desired. The dimensioning help is also available in IAMs when a part is in edit mode and the sketches have been switched on.

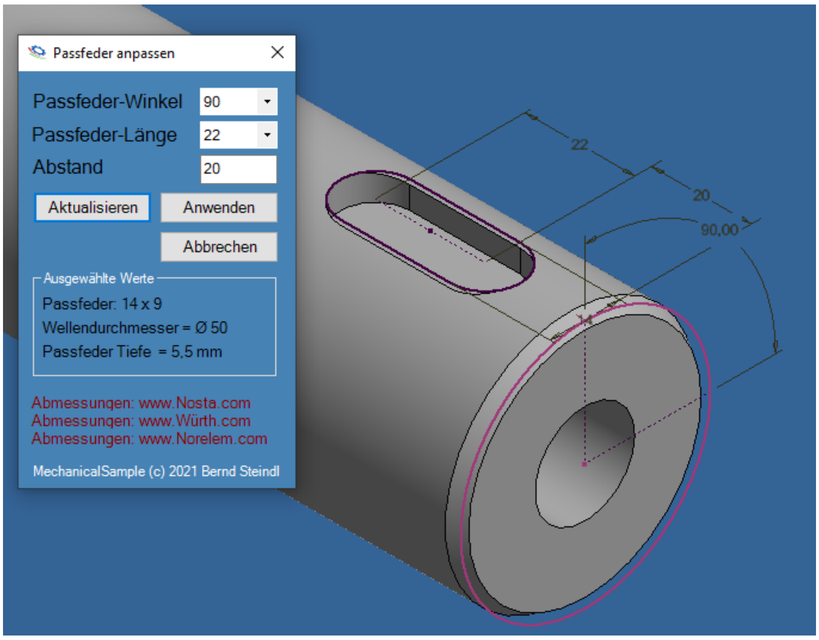
Punctures (The puncture tool can only be used in IPT's).



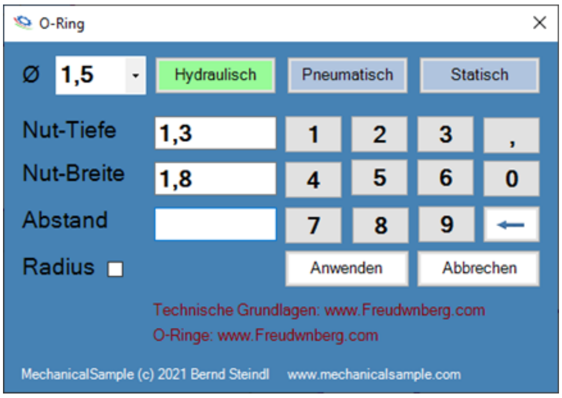
This tool can be used to make recesses for O-rings and retaining rings in rotationally symmetrical workpieces or in holes in cuboids. A direct call of proven iFeatures for thread undercuts and undercuts on turned parts is possible. When an iFeature is started, the working planes for the selection are automatically switched on and switched off again when the iFeature is finished.

The key tool can be used to create keyways on shafts and in bores according to DIN471 and DIN472. It is sufficient to select the plane start surface and the desired diameter. During generation, the key length, the distance from the planar surface and the angle on the shaft or in the bore can be freely selected. The key width is automatically selected according to DIN 6885. On the dialog there are direct links to the different key manufacturers. The web pages are opened in your standard browser.

Feather key dialog for external or internal feather keys

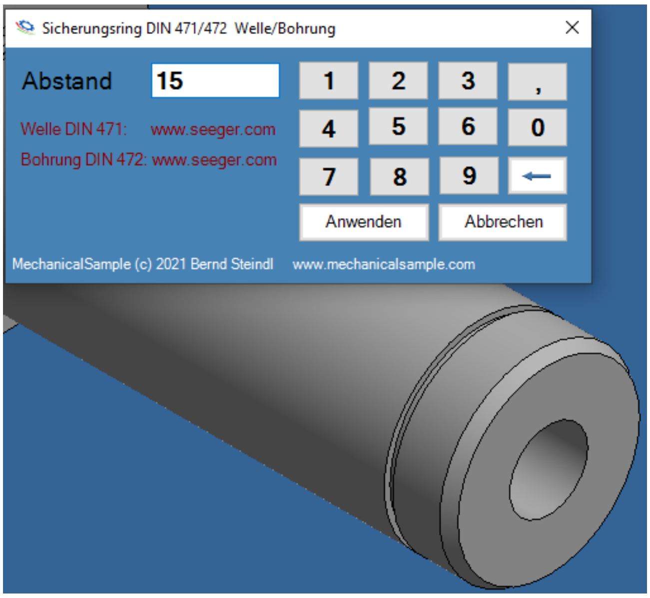


O-Ring Dialog



When selecting the "O-Ring Button" another dialog opens to specify the dimensions of the O-Ring groove. Cord thickness, hydraulic, pneumatic and static as well as the distance from the selected planar start surface must be selected. On the dialog there are direct links with technical basics about O-ring grooves. The web pages are opened in your standard browser.

Retaining -Ring Dialog



When you select the "Retaining ring button", another dialog opens to create the recess for a retaining ring. Retaining rings can only be created on integer diameters

Sketches icon

With the sketch symbol all sketches are switched on. For small components with a low density of sketches, it can be advantageous to switch on all sketches with a single click. With the 2nd click on the same symbol all sketches are switched off again.

Switch on workplanes icon

(Attention: The sketches and Workplane symbols do not have the same function in IAM's!)
The layer icon fulfills several options by multiple click.
With the 1st click only the origin Workplanes are switched on without the user Workplanes in components. If there are only origin Workplanes, they will be switched off by the 2nd click on the symbol. If user Workplanes are available, they are added to the origin Workplanes by the 2nd click on the symbol. With the 3rd click, the origin Workplanes are switched off so that only the user origin are visible.With the 4th click all Workplanes are switched off

Clean icon

Clean icon turns off all sketches and all Workplanes

The section symbols

YZ -Schnitt

XZ -Schnitt

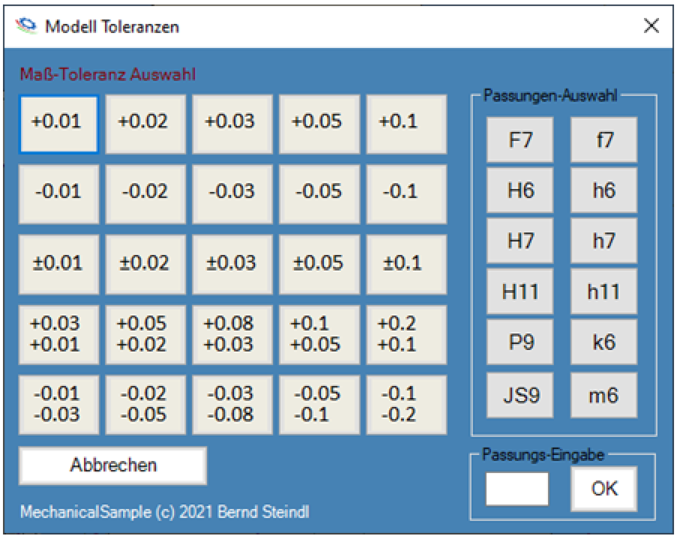
XY -Schnitt

The part will be cut directly to the selected origin Workplanes

Tolerances in IPT models

Toleranzen

This tool allows to generate a preselection of tolerances on dimensions in models.



The tolerances can be retrieved in the IDW by "Retrieve model annotations" and thus be generated in Idw's. Model tolerances behave associatively to the referenced IDW file. The associativity can be switched off in the MechanicalSample options. Inserting tolerances in models allows to send dimensioning information to suppliers without using an Idw. Individual fit in put according to the Inventor® fit list is possible.

Create Step File

this tool puts a step file into the folder C:/temp. After the creation the folder opens for further use of the file. The file should be deleted afterwards to create another step file on this path.

Colorize holes

This tool allows to color the first hole of a rectangular or round arrangement. This is often helpful to find the first hole when placing components in arrays. The tool is for users whose company does not yet have a regulation regarding hole coloring.

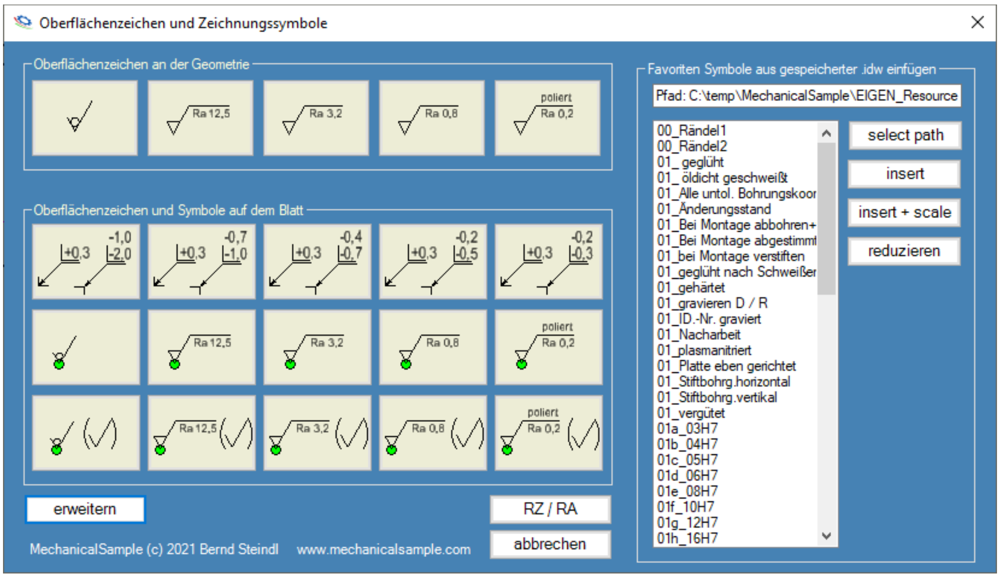
1.5 Commands for IDW's

Surface characters and drawing symbols

This tool offers the most important surface symbols for direct insertion on the sheet or on a geometry.

The dialog can be switched from RA to RZ. When inserting at a dimension, the tool recognizes a linear dimension and creates the symbol at both dimension lines simultaneously. The tool also offers a quick insertion of edge break symbols according to DIN ISO.

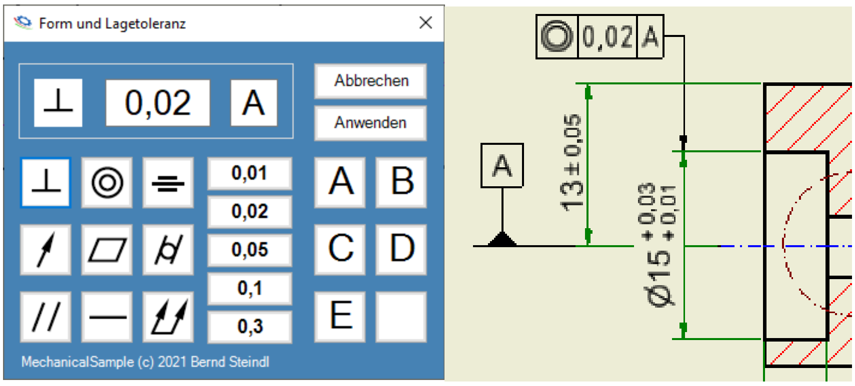
The extension mode:



To place the most important own or company symbols directly on the drawing you can extend the dialog. To do this, open an IDW drawing file and place all the desired symbols in this drawing at any position and save them in any folder. Then the "select Path" button can be used to find this folder and the IDW. In the selection window, the corresponding symbols appear, which can then be inserted directly into the drawing by selecting them and clicking on the "insert" button. A special feature is the "insert + scale" button. This button allows you to insert and place the symbol in case of different scaling of the views in an IDW. Then, by clicking on the scaled view, the symbol is scaled with the scaling of this view.

Shape and position tolerances

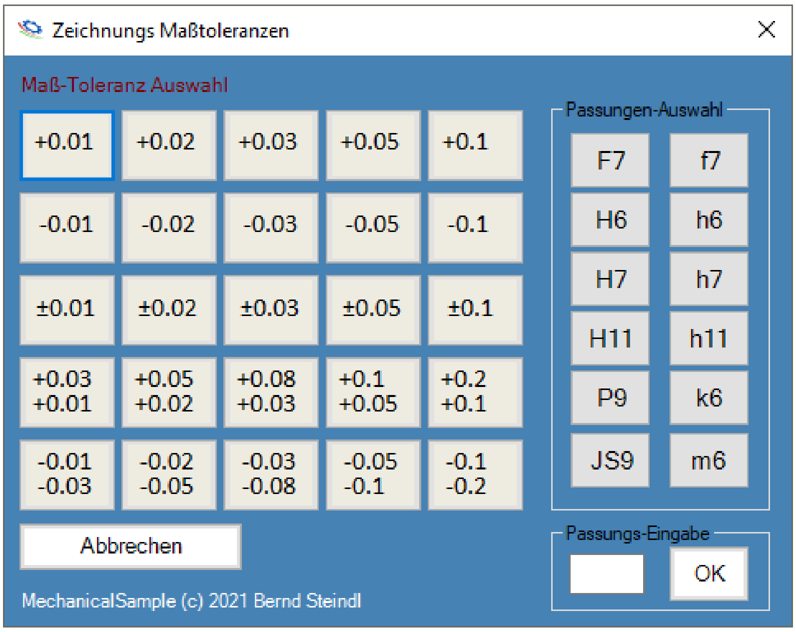
This tool allows to create predefined, or by individual input into the tolerance field, shape and position tolerances directly at the dimension line. The symbol is placed according to the selected dimension line directly at the dimension arrow. If the symbol is required on a geometric contour, it can be created on a dimension line and the arrowhead can be dragged to this contour.



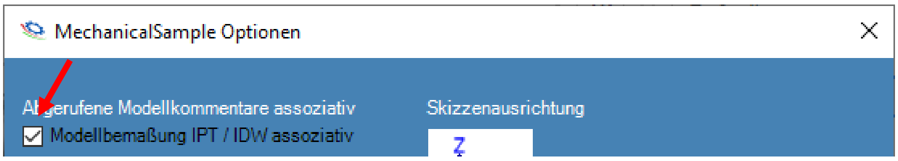
Tolerances in IDW drawings

Toleranzen

This tool allows to create a preselection of tolerances on dimensions in IDW dimensions. The values of fit tolerances, e.g. H7, are displayed with dimensions. Mechanicalsample offers the possibility to place a fit table on the drawing, in which the fit tolerances can be changed to <Linear> or <Stack> after creating the table. See below under Fit Table.



Dimension tolerances always behave associatively to the referenced IPT file when the dimension is retrieved from the model. Associativity can be turned off in the MechanicalSample options. No associativity to existing model dimensions will be established if the model dimension was retrieved in the IDW. The setting can be found in the MechanicalSample options "Model dimension IPT/IDW associative".



Diameter characters

Ø

In some cases Inventor® does not generate an Ø symbol when creating a dimension. In this case, by clicking the button and then selecting the dimension, the symbol can be quickly inserted.

Add dimension text

x30

The dimensioning of a chamfer can be supplemented with dimension x30°.

Add dimension text

x45

The dimensioning of a chamfer can be supplemented with dimension x45°.

Equal characters

=

The symbol allows to replace the selected dimension with an equal sign.
The command is called 2x in a row to create a symmetry sign.

Pitch circle sign with tolerance

Tk.

Completes the dimension text with the pitch circle designation and a tolerance +/- 0.1

Bracket character

(x)

Sets the dimension text in brackets

Centerlines Symbol

Creates the center lines in all views of the IDW

Thread Symbol

creates the thread lines in all views with threads.

Zoom Symbol

Zoom to sheet size of the current IDW sheet

PDF Symbol

Creates a PDF file

DXF Symbol

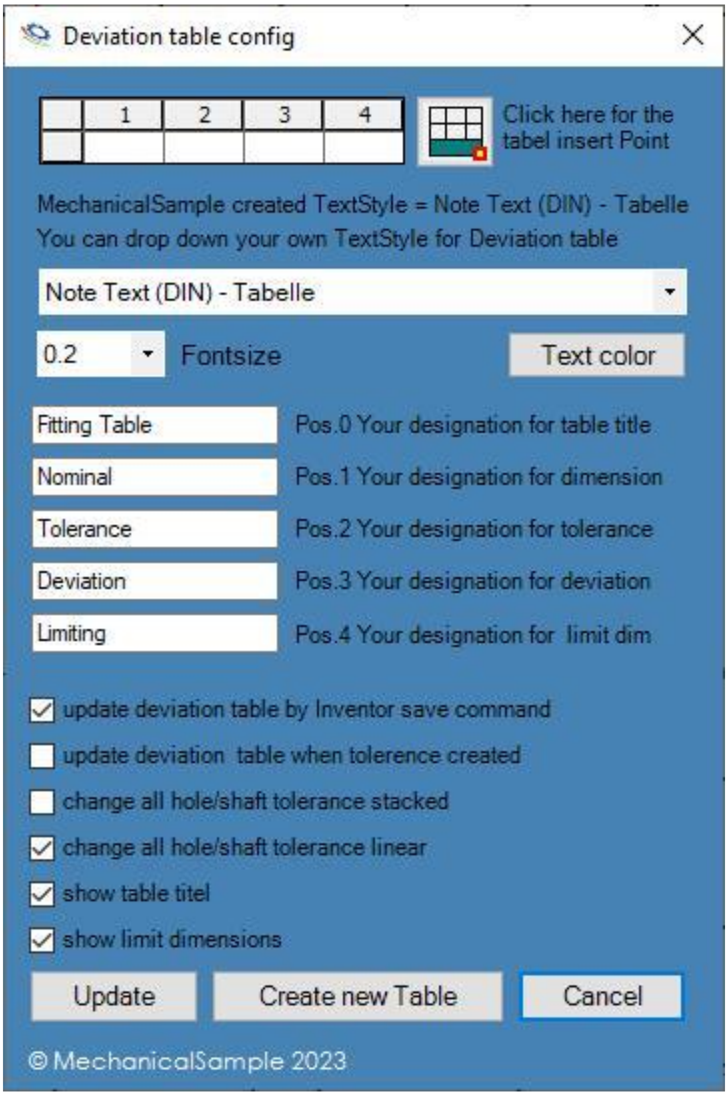
Creates a DXF file

Tabel Symbol

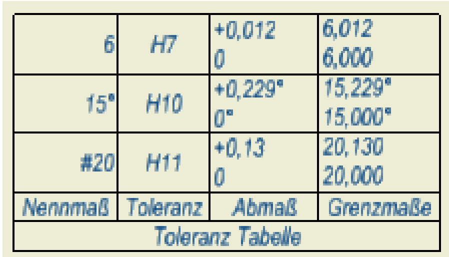
Fit table:

With the fit table, drawing tolerances can be inserted in table form at any position that can be determined with the mouse. The insertion point can be set at the bottom right, top right, top left or bottom left, as desired. Text style, text size and text color can be set individually according to company specifications. Likewise the designation of the table title and the table rows 1-4. Dynamic adjustment of the table view to the dimension values.

The option: Update with Inventor® Save command  ensures that the table updates automatically when the drawing is saved. This means that no tolerance is "forgotten" anymore Optionally, a limit dimension table can be switched on.
An automated setting of all tolerances to a uniform tolerance format can be created with the options: "Change tolerance display to stack view" or "Change tolerance display to linear view" during the creation of the table.
If overwritten dimensions are present in the drawing and they have a tolerance, the overwritten dimensions are marked with a # (hash) in the table. Thus, dimensions that are not to scale are recognizable in the table.

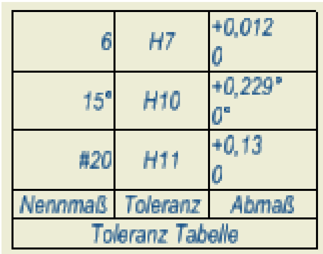


View: limit dimension + table designation



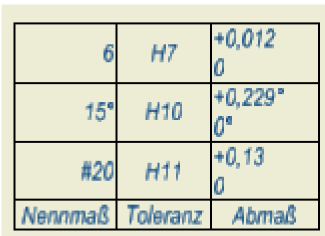
6	H7	+0,012 0	6,012 6,000
15°	H10	+0,229° 0°	15,229° 15,000°
#20	H11	+0,13 0	20,130 20,000
Nennmaß	Toleranz	Abmaß	Grenzmaße
Toleranz Tabelle			

View: With table designation



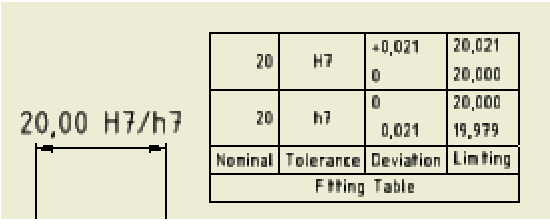
6	H7	+0,012 0
15°	H10	+0,229° 0°
#20	H11	+0,13 0
Nennmaß	Toleranz	Abmaß
Toleranz Tabelle		

View: Minimal

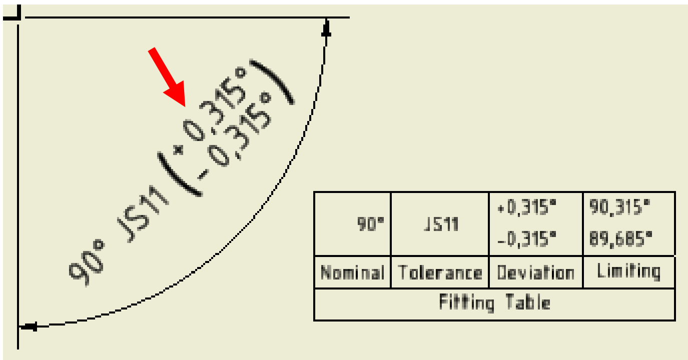


6	H7	+0,012 0
15°	H10	+0,229° 0°
#20	H11	+0,13 0
Nennmaß	Toleranz	Abmaß

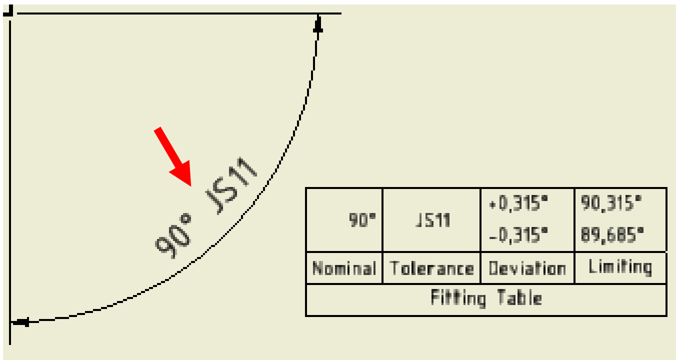
Processes stack and linear tolerances



Processes angle tolerances



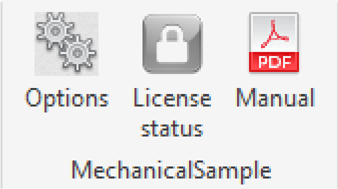
Sets all tolerances to stack or linear during the generation of the fit table



Processes tolerances of overridden dimensions and marks them with #.

90°	JS11	+0,315° -0,315°	90,315° 89,685°
#200	H11	+0,29 0	200,290 200,000
Nominal	Tolerance	Deviation	Limiting
Fitting Table			

1.6 License.lic – File



MechanicalSample requires the Licence.lic file included in the download package.
The Licence.lic file must be located in the folder:

C:\temp\mechanicalsample\

so that MechanicalSample can work. Copy the file into this folder.
You must have read and write permission for this folder. By clicking on the "License Status" icon on the start screen, the user can check the status of his license.
Autodesk versions Inventor ® every year. For this reason, the duration of mechanicalsample is usually set to one year. After this period, a new mechanicalsample package must be downloaded from
https://mechanicalsample.com/en/en-download/
and distributed according to the folder defaults to keep mechanicalsample up to date with corresponding newer Inventor ® versions.

Lizenz-Status-Dialog