

3DViewStation WebViewer Tutorial

Best Practice



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The logo for KISTERS, featuring a stylized 'K' symbol followed by the word 'KISTERS' in a bold, sans-serif font.

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Chapter I:

3D Data Analysis and Editing Functions



1 3D Data Analysis and Editing Functions

In this chapter you will find detailed descriptions of these functions:

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1.1 Model Orientation

Various functions are available for orienting the model. After opening a model, the two functions **ZOOM RECTANGLE** and **SELECT** are activated by default. You control both functions using the left mouse button.



The **Select** [6](#) function is executed by clicking the left mouse button. You can change the function by clicking on the left mouse button in the **START** tab > **SELECTION MODE**, for example.

The zoom rectangle, which is activated as well, can also be controlled with the left mouse button: If you hold down the left mouse button, you can draw a frame in the model window. This function allows you to adapt the display area to the content of the frame. The function with pressed left mouse button can be changed in the **START** > **OPERATION** tab.

The model area is adjusted to the display of all objects by double clicking in an empty area of the model pane or via **ZOOM** in combination with the function **FIT**.


Turning the mouse wheel or using the function **ZOOM IN** or **ZOOM OUT** will increase or reduce the display size.

Clicking and holding the right mouse button while moving the mouse within the model area will rotate the view. The virtual camera is moved around the model, while the model remains static. The rotation point is located either along the center line of the model or (with the right mouse button clicked on a particular spot of the model) along the cursor line. This rotation point is visualized with a cross.

A single right click of the mouse opens the context menu and allows quick access to frequently used functions, which will be explained in more detail in the coming chapters.

By pressing both mouse buttons or the middle mouse button (mouse wheel) and simultaneously moving the cursor, the view of the model can be moved.

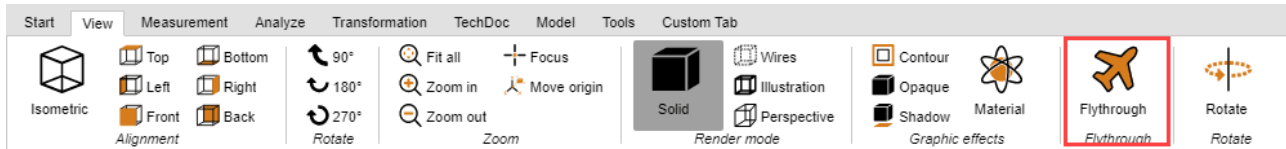
You can change more mouse settings and pre-assignments via **SETTINGS** > **DEFAULT** > **SCENE** > **MOUSE**.


More options for the orientation and visualization of the model are described in chapter [Customizing the View](#) .

1.2 Flythrough mode

With the flythrough mode you can fly through any geometry with the camera, for example to measure the inside of buildings or to gain an impression of the object's proportions.

The **FLY THROUGH** function can be found under the menu item **VIEW > FLY THROUGH**.




When activating the flythrough mode, the render mode **PERSPECTIVE** is activated first (see also chapter [Customizing the View](#) ). Due to the required perspective representation, the distance to the geometry is first adjusted. To avoid large jumps of the camera, the perspective can be activated first under **VIEW RENDER MODE**. At the same time, the up vector (defines the top orientation) is set according to the value set under **CAMERA UP VECTOR** and remains constant for the course of the flythrough mode in order to prevent the camera from being tilted.


Controls

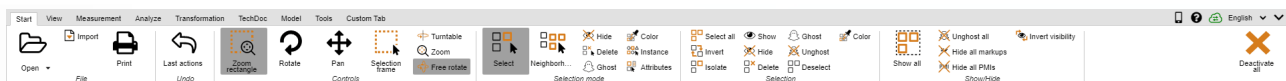
The camera can be moved using a keyboard, mouse or 3Dconnexion 3D mouse.

- Use the **[W]** / **[A]** / **[S]** / **[D]** keys or the arrow keys to move the camera forward or sideways.
- The **[SPACE BAR]** / **[C]** or the page up and down keys are used for up and down movement.
- By holding down the left or right mouse button without moving the mouse, the camera flies forwards or backwards, whereby the direction of movement of the camera is adjusted by subsequent mouse movements.
- By pressing the right mouse button and moving the mouse, the viewing direction of the camera can be adjusted as usual. The delay for triggering the **FLYING WITH PRESSED MOUSE BUTTON** can be adjusted in the **DELAY WITH PRESSED MOUSE BUTTON** option.
- The **[+]** / **[-]** keys adjust the speed at which the model is flown through.

To obtain information on the individual settings, click on a setting in the tool area. You will then receive detailed descriptions of the respective setting in the lower area of the windows – in the so-called Help Box. When creating [views](#) , the settings made for flythrough mode including camera position are maintained. When leaving the flythrough mode, the camera is reset to the last known position.

1.3 Object Selection

Many advanced tools require a previous selection. Selections are created via the following functions and are saved as described in chapter [Saving Selections](#) .



Individual parts of an assembly can be selected by clicking them directly within the model area or by clicking the corresponding entry in the model structure for more processing steps.

The selection can be cancelled by pressing the **[ESC]** key or by clicking in the background of the model area.

With **[CTRL]** pressed at the same time, more geometries can be added to the selection. Clicking the selected geometries again will deselect them.

With the **[⇧]** key pressed at the same time in the model structure, all parts can be selected which are grouped in the structure between the first and the last selected entry.

Clicking an object will initially select the relevant geometry (lowest level in the **STRUCTURE** view). A further click of a previously selected object selects the next higher level of the structure, and so on until the entire model is selected. Another click will then return the selection to the initial geometric element. This enhanced selection can be disabled via **SETTINGS > SCENE > GENERAL**.

You can also choose a Selection frame via **START > CONTROLS**. It can be used by dragging a pane with the left mouse button pressed. The content inside the frame will then be selected.

Please note that all geometries that at least partially lie within the selection frame are selected when the frame is drawn from left to right. Dragging the pane from right to left on the other hand, will only select geometries completely contained within the frame.

All visible geometries can be selected by clicking the **[SELECT ALL]** button or by using the shortcut **[CTRL]+[A]** in the **SELECTIONS** area.

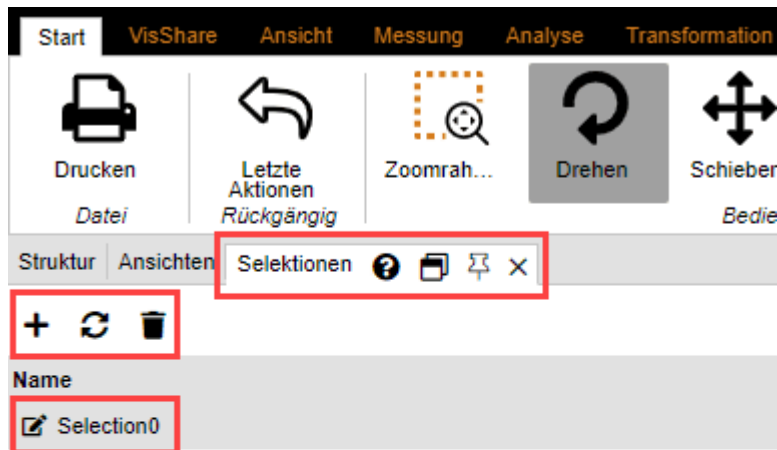
You will find a description of other options for creating specific selections in ch. [Filter \(Search\)](#)⁴⁴.

1.4 Saving Selections

3DViewStation offers you the possibility to save selected objects of an assembly in selections.


Saving a Selection

- On the **VIEW** tab, from the **SHOW PANES** function group, select **SELECTIONS**.
- Select one or more objects of an assembly. cf. [Object Selection](#)⁶⁴
- In the **SELECTIONS** pane, click the **+** icon.




The selection is created as a list entry with the name **SELECTION0**.


Rename selection

- In the sub-window, select the **SELECTIONS** tab.
- Select the desired selection from the list entries and hold down the left mouse button for one second. Alternatively, click the  icon to activate the edit mode.

Update selection

- In the sub-window, select the **SELECTIONS** tab.
- Select the desired selection from the list entries.
- While holding down the **[CTRL]** key, click on any number of additional objects in the assembly. Re-selecting objects that have already been selected will cancel the selection.
- Click the  icon to update the selection.

Delete selection

- In the sub-window, select the **SELECTIONS** tab.
- Select the desired selection from the list.
- Click the  icon to delete the selected selection.

Tip



- Just like for the selection of multiple geometries, you can press and hold the **[CTRL]** key at the same time to combine various saved selections.
- Selections are saved with the 3DVS format when saving.

1.5 View and Edit Properties of Selected Objects

Once an object is selected, the sub pane **PROPERTIES** displays the node properties and can be edited there (see first/left image):

Where multiple objects are selected, only those properties will be displayed under **NODE PROPERTIES** that all objects have in common. Otherwise the entry **MISCELLANEOUS** will be displayed (see second/right image):

Properties ? [Print] [Refresh] [Close]

↑↓

[-] Node properties

[-] General

Name	MechanicalTool.1
Type	Ri_BrepModel
Node ID	323
Geometry ID	22
File path	N/A
Contains geometry (BREP)	<input checked="" type="checkbox"/>
Material	None ▼
Color overwrite	Vertex ▼
Color	<input type="color" value="#00FF00"/>
Transparency overwrite	Node ▼
Transparency	0 <input type="range" value="0"/>

[+] Position bounding box center

[+] Bounding box extents

Compute

[-] Physical properties

Surface	N/A
Volume	N/A
Density	N/A
Density preset	▼
Mass	N/A

[-] Gravity center

X	N/A
Y	N/A
Z	N/A

Create gravity center

[+] Render states

[+] Animation

[+] Constraints

Properties ⓘ 📄 📌 ✕

↑↓

[-] Node properties ▲

[-] General

Name	MechanicalTool.1
Type	Ri_BrepModel
Node ID	__VARIOUS__
Geometry ID	__VARIOUS__
File path	N/A
Contains geometry (BREP)	<input checked="" type="checkbox"/>
Material	None ▼
Color overwrite	▼
Color	XXXXXXXXXX
Transparency overwrite	Node ▼
Transparency	0 <input type="range"/>

[+] Position bounding box center

[+] Bounding box extents

Compute

[-] Physical properties

Surface	N/A
Volume	N/A
Density	N/A
Density preset	▼
Mass	N/A

[-] Gravity center

X	N/A
Y	N/A
Z	N/A

Create gravity center

[+] Render states

[+] Animation

[+] Constraints

Changes applied to multiple selected objects (e.g. color) for which **MISCELLANEOUS** is displayed, will affect all selected objects.

Clicking the background will deselect an existing selection and displays the properties of the current scene. You can also find these scene properties under **SETTINGS > CURRENT SCENE**, a pre-configuration for new scenes can therefore also be done via **SETTINGS > DEFAULT > SCENE**.

If the **PROPERTIES** pane has been hidden by another pane, creating a selection brings the properties pane to the foreground. This behavior can be deactivated via **SETTINGS > DEFAULT > APPLICATION > UI > ACTIVATE PROPERTIES WINDOW**.

1.6 Selection modes

Clicking **HIDE**, **DELETE** or **GHOST** in the **SELECTION MODE** tab of the **START** ribbon will hide (temporary), delete (permanent) or ghost (see chapter [Ghost Mode](#)¹³) all with the mouse selected objects, until the relevant function is deactivated by pressing [**ESC**] or clicking the **SELECT** function.



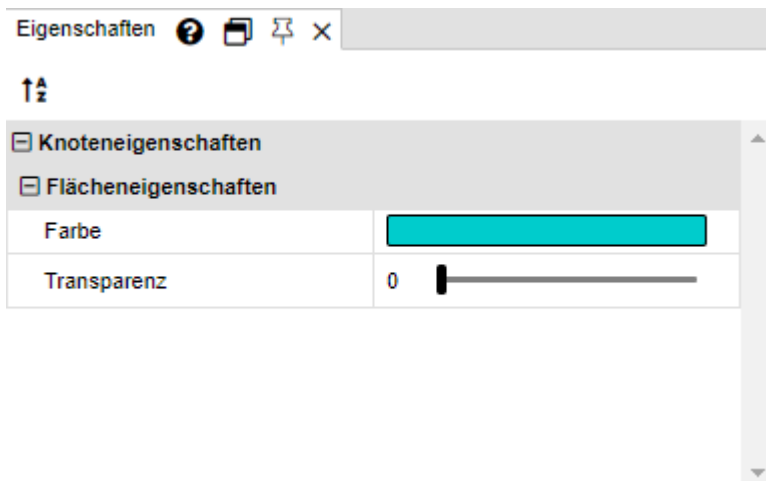
Color

The selection mode **COLOR**, allows you to select all objects of a certain color in an assembly.

Surface

The selection mode **SURFACE**, allows you to select individual surfaces of objects. After selecting a surface, hold down the [**CTRL**] key to select multiple surfaces.

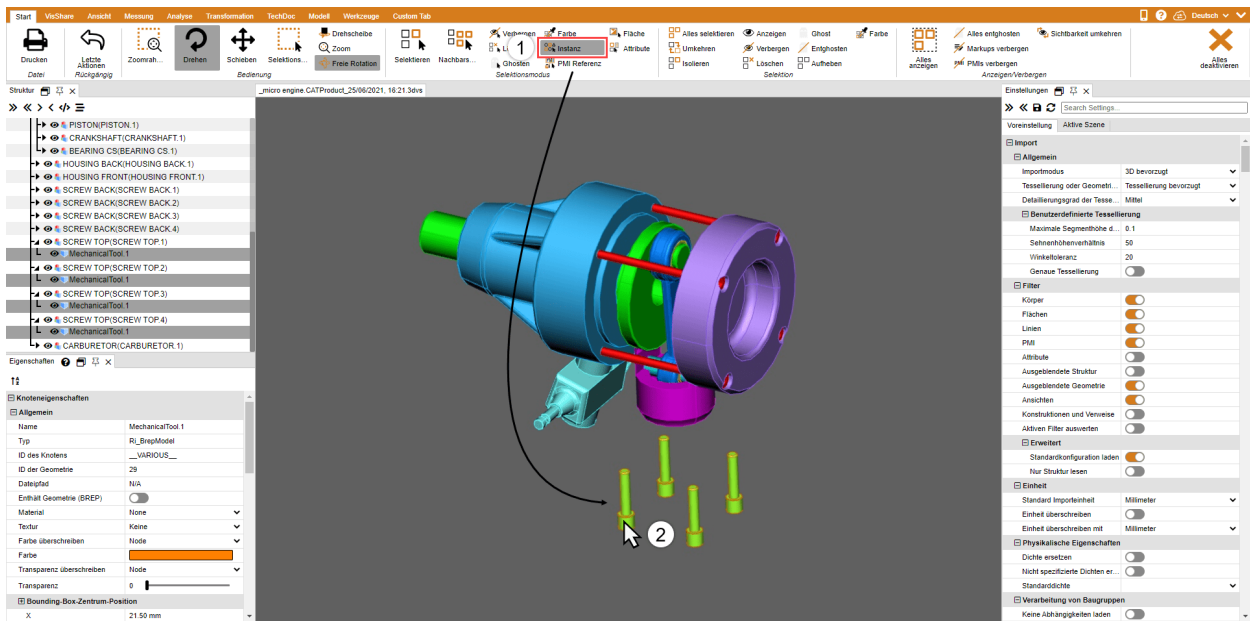
In the info area, you can adjust the color and transparency of the selected surfaces in the surface properties.



Instance

Identical objects are treated as a single instance by 3DViewStation. You can use the **INSTANCE** selection mode to select these object groups.

- On the **START** tab, in the **SELECTION MODE** area, activate the **INSTANCE** mode.
- Select an object that exists in identical form several times in the assembly.

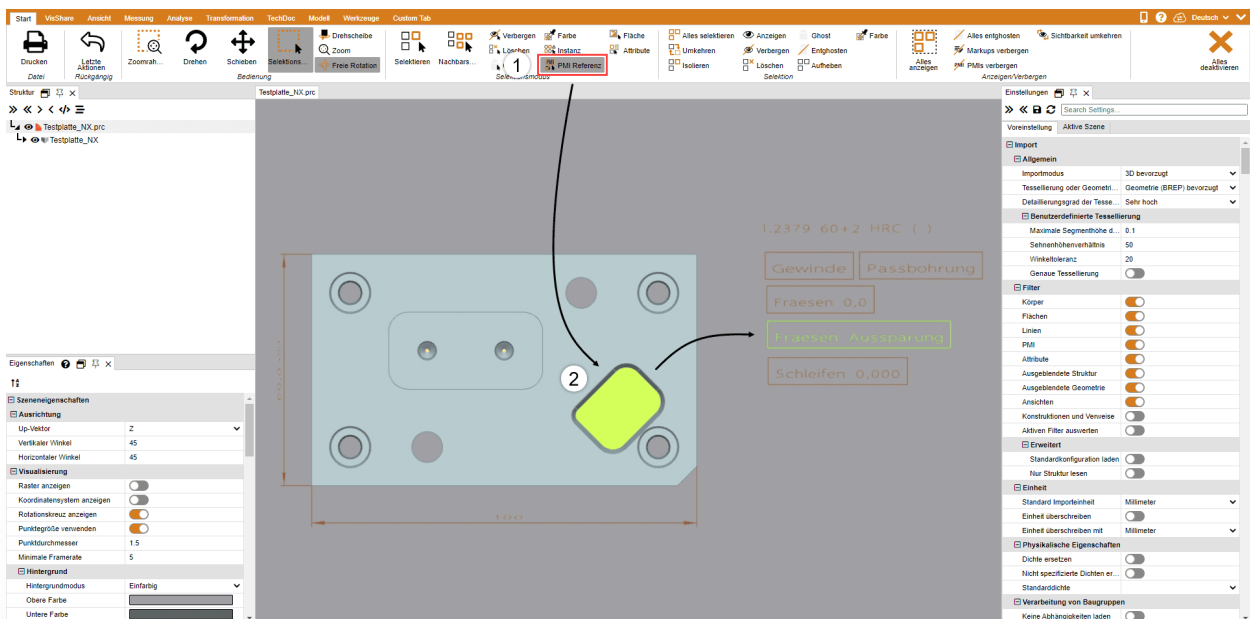


All identical objects that match the original selection will be selected.

PMI Reference

The **PMI REFERENCE** selection mode allows you to display the PMI reference (product manufacturing or design information) that matches a surface.

- On the **START** tab, in the **SELECTION MODE** area, activate the **PMI REFERENCE** mode.
- Select a surface in the model.



The PMI reference associated with the surface (if it exists) is highlighted in color.

1.7 Apply Functions to a Selection or to All Objects

In addition to the direct selection mode, previously selected objects can also be hidden, isolated or deleted via the respective buttons in the **START > SELECTION** area. The functions in the **SHOW > HIDE** area are always applied to all objects.

Some of these functions can also be accessed via the right-click context menu or via a shortcut.

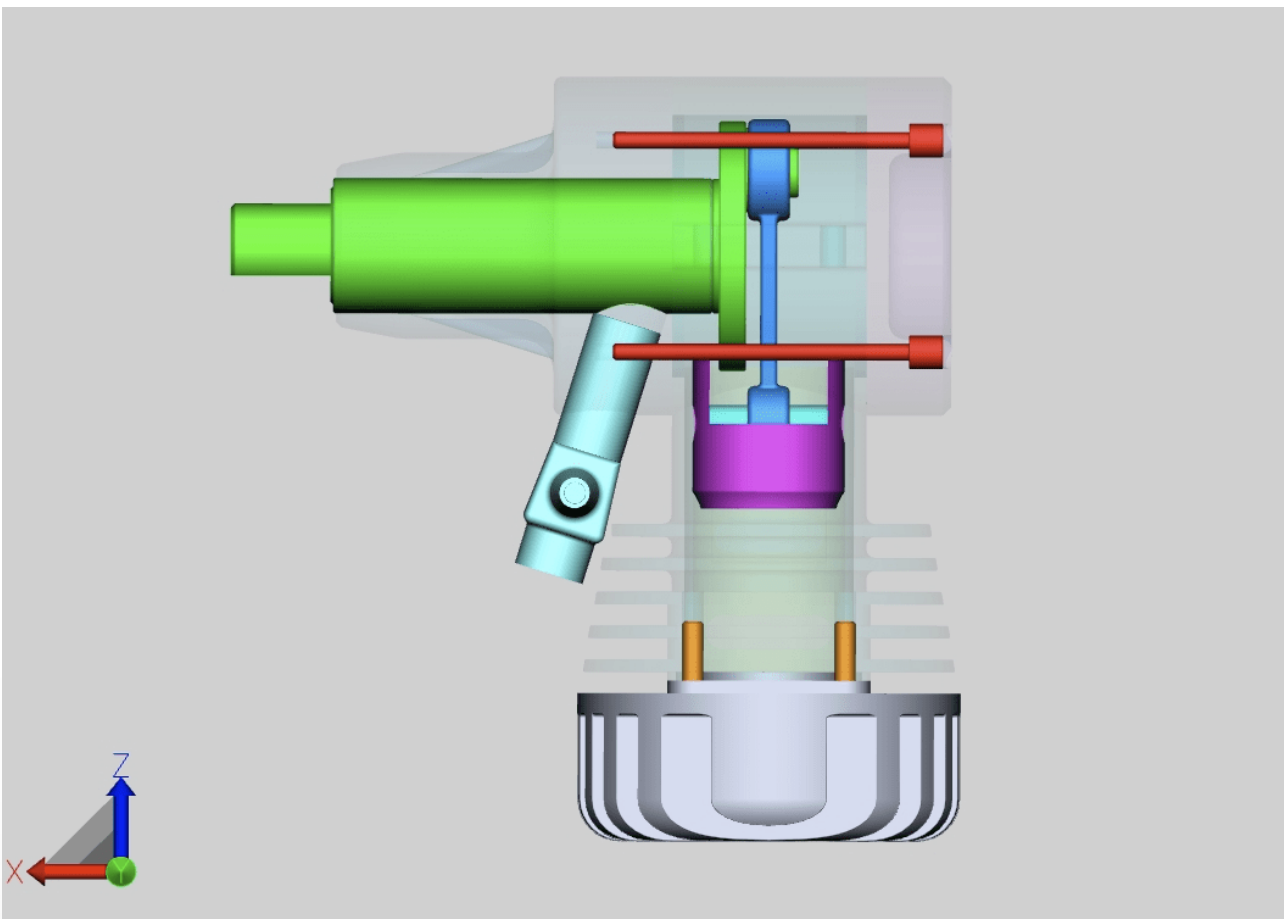


1.8 Ghost Mode

GHOST mode gives parts transparency, but unlike purely transparent geometries, ghosted geometries in the model area can no longer be selected. That means you can select and dimension the part behind the ghosted part. These ghosted geometries are displayed with a grayed out icon in the structure.

Enable the ghosting mode via **START > SELECTION MODE > GHOST** or select the relevant objects and click the button **START > SELECTION > GHOST**. Alternatively you can use the key combination **[ALT]+[G]** to ghost the object at the cursor position (no previous selection required).

To make all ghosted parts visible again, the **UNGHOST** function can be selected in the **SHOW > HIDE** area. The **GHOST** mode is also disabled by selecting the **SHOW ALL** function. The key combination **[ALT]+[S]** will unghost all parts one by one in reverse order. A selective deactivation of the **GHOST** mode is made possible by selecting corresponding geometries in the structure and then selecting the **UNGHOST** option in the **SELECTION** area.



1.9 Neighborhood Search

The neighborhood search finds all geometries whose bounding box intersects the sphere radius specified during the search by the current selection.

Neighborhood search for an object of an assembly

- Initiate the neighborhood search with the **NEIGHBORHOOD** button from the **START** tab in the **SELECTION MODE** area and click on the desired geometry or create a selection. Alternatively, you can first select the geometry and then click the **NEIGHBORHOOD SEARCH** button.

The **NEIGHBORHOOD SEARCH** pane appears in the right-hand tool area, where you can edit the search radius or the search distance. Initially, the dimensions of the Bounding Box of the selected original geometry is used as the search radius.

- Returned geometries can then be selected and/or isolated depending on the desired option. You can also include hidden geometries.

For more information on the individual options of the neighborhood search, please refer to the tooltip of the respective option.

Nachbarschaftssuche ? 📄 📌 ✕

☐ Nachbarschaftssuche

☐ Modus

Form	Box	▼
Verhalten	Einzel	▼
Auf alle anwenden		

☐ Selektiere

Einschließend	<input checked="" type="checkbox"/>
Schneidend	<input checked="" type="checkbox"/>
Ausschließend	<input type="checkbox"/>

Die gefundenen Geometrien werden selektiert.

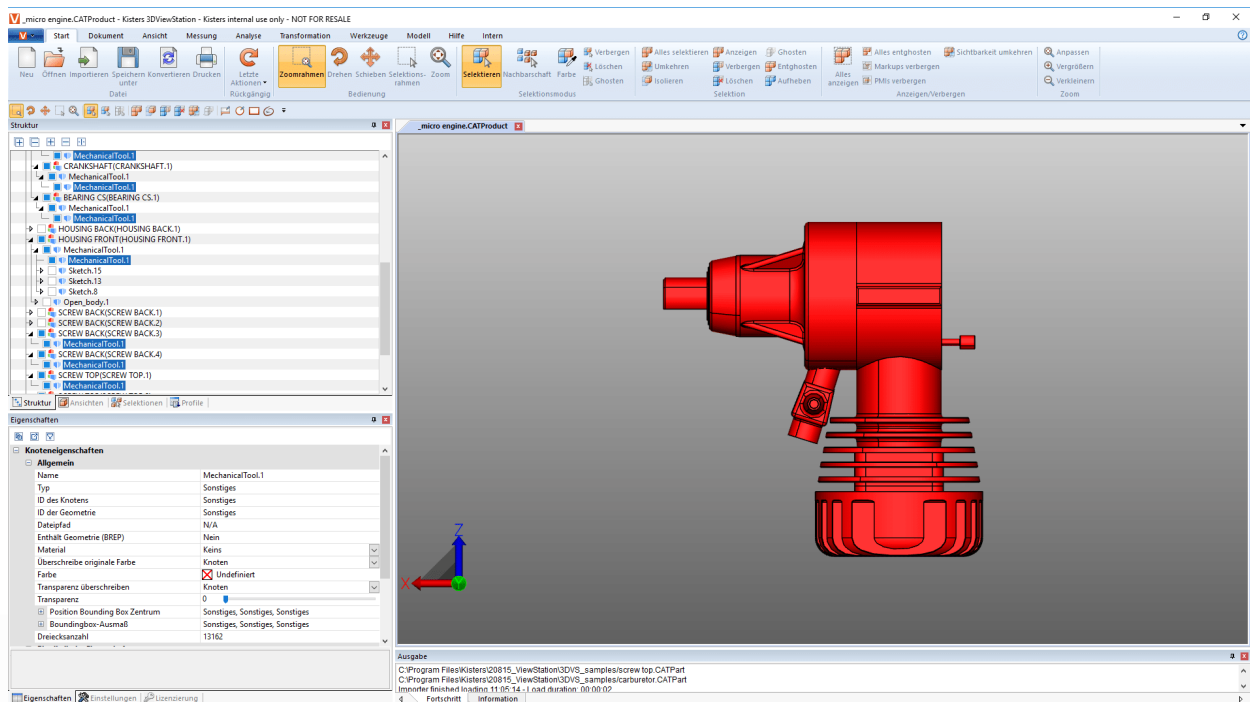
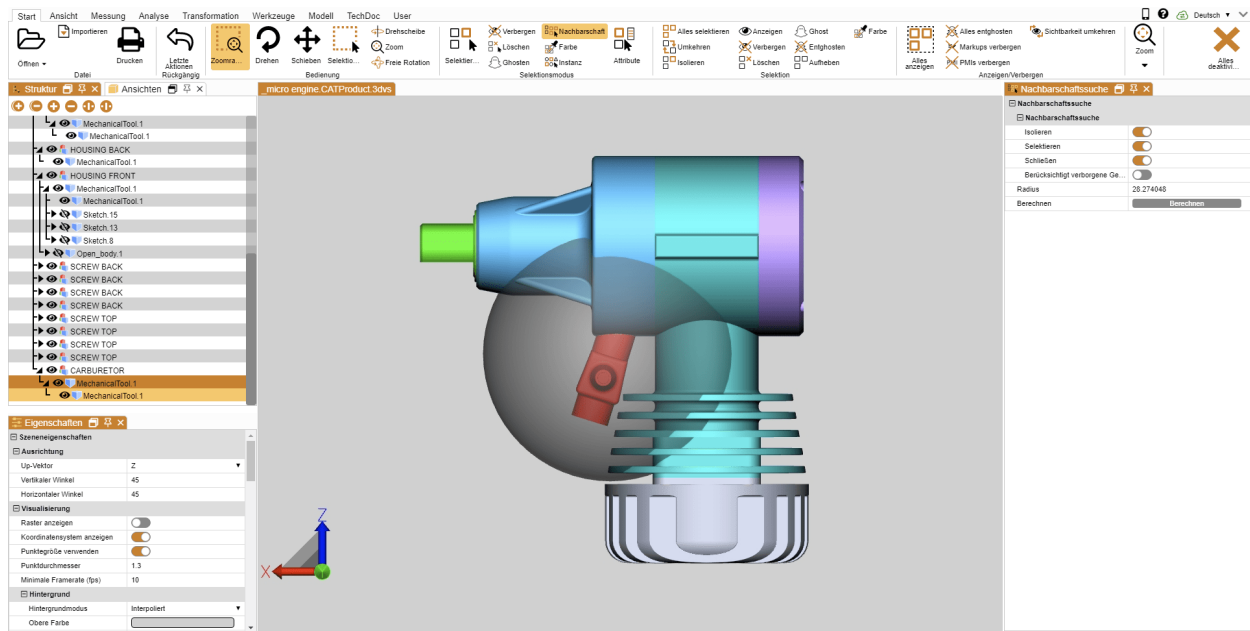
Selektieren	<input checked="" type="checkbox"/>
-------------	-------------------------------------

☐ Einstellungen

Schließen	<input checked="" type="checkbox"/>
Versteckte Geometrien berück...	<input type="checkbox"/>
Nur Bounding-Box testen	<input type="checkbox"/>
Form sperren	<input type="checkbox"/>
Farbe	<input style="width: 100%;" type="text"/>


Berechnen

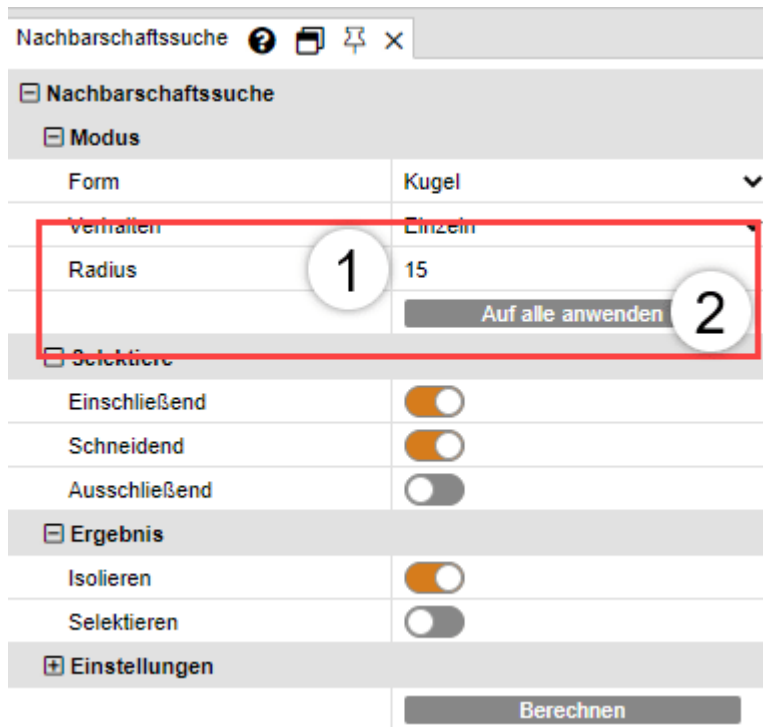
- Start the search by activating the **[COMPUTE]** button.



Neighborhood search for multiple objects of an assembly

You can also apply the neighborhood search to several selected objects at the same time.

- Start the neighborhood search with the **NEIGHBORHOOD** button from the **START** tab in the **SELECTION MODE** area.
- Click on a desired object in the assembly.
- While holding down the **[CTRL]** key, click on any number of additional objects in the assembly. See [Object Selection](#) 
- Step 1: In the Neighborhood Search tool area, adjust the search radius or distance.
- Step 2: Click the **APPLY TO ALL** button.



- Start the search by activating the **[COMPUTE]** button.

1.10 Customizing the View

The **VIEW** tab offers further visualization options.



The isometric view is preconfigured with a 45° vertical and horizontal angle. The configuration of these angles can be changed via **SETTINGS > DEFAULT > SCENE > CHANGE ORIENTATION** or those of the current scene by clicking the background under **PROPERTIES > SCENE PROPERTIES > ALIGNMENT** (see chapter [View and Edit Properties of Selected Objects](#) ^[8]).

In the **Zoom** area, the **Focus** function allows you to center a selection (see chapter [Object Selection](#) ^[6] and [Saving Selections](#) ^[7]) in the Model Area. This function is useful to focus on a node selected in the structure (e.g. sub assembly). This function is also available in the context menu (right click).

The functions **RENDER MODE** and **GRAPHIC EFFECTS** adjust the scene of the current model. You can, for example, display the wireframe without fill, switch to monochrome rendering of the model in the Illustration mode or click the **OPAQUE** button to temporarily remove the transparency of all geometries. In other words: these are simply display effects, which have no effect on an export.

You can also use the option **SHADOW** to add a shadow thrown by the part. The shadow behavior can be influenced via **PROPERTIES > SCENE PROPERTIES > SHADOW**. Some effects will require a recalculation of the shadow. This can be done in one pass by exiting the shadow and then reactivating it, or continuously by activating the setting **REAL-TIME CALCULATION**.

Under **SHOW PANES** you can hide or show panes such as the **SETTINGS** pane. You can also reset the entire user interface to its default settings; this action will not affect settings.

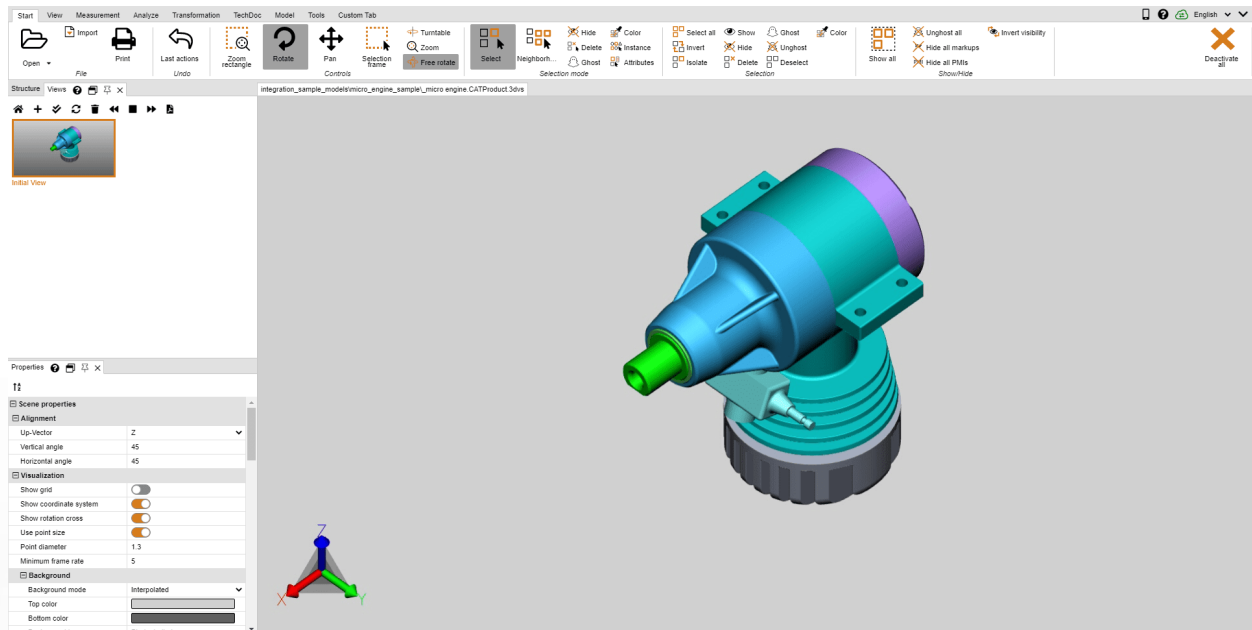
1.11 Views

Specific work steps can be captured by generating a view of the current scene via the sub pane **VIEWS** in the **STRUCTURE** area.



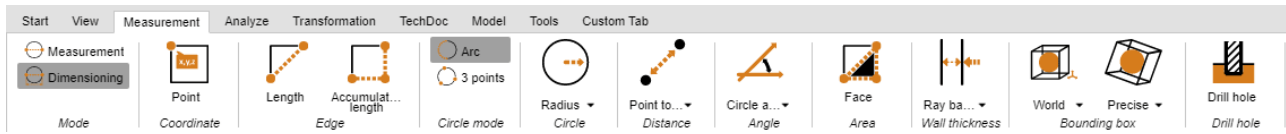
Function	Description
START	Enables the view displayed at last save.
CREATE	Generates a view from the current scene.
UPDATE	Overwrites the selected view with the current scene.
DELETE	Deletes the selected view(s).
PLAY	Starting with the selected view, the previous (Reverse play) or the subsequent (Play) views will be displayed.
STOP	Stops the playing of views.


- Double clicking a view restores this scene.
- A view can be repositioned via click & drag.
- Left-click on the name of the selected view to rename it.







1.12 Measurement

The following steps are required for measuring a model without annotation. Please read chapter [Dimensioning](#) for measurements with annotations:



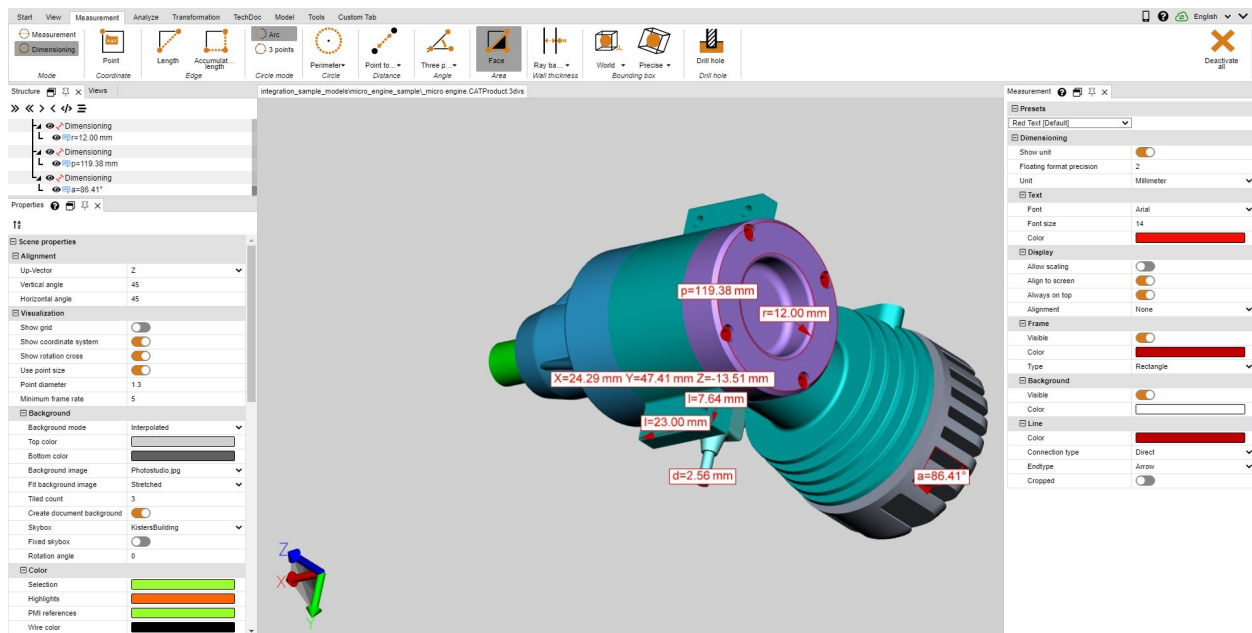
- Enable the **MEASUREMENT** mode in the **MEASUREMENT** tab and select one of the measurement functions.
- Place the cursor on the geometry you wish to measure. Depending on the chosen measurement function, the reference object found will be highlighted as a suggestion. Left clicking it will select that reference object for the actual measurement.
- Additional operating instructions are displayed in the lower part of the tool pane. All other navigation functions, e.g. zoom (= mouse wheel), view rotation (right mouse button), or view move (press and hold middle mouse button) remain available to navigate to other objects.
- Measured values are displayed in the **MEASUREMENT** sub pane.
- Clicking the relevant button in the function area again will end the measurement function, as will clicking  in the tool pane or pressing **[ESC]**.

Measurement    	
Presets	
Red Text [Default] ▼	
Point measurement	
Show x-coordinate	<input checked="" type="checkbox"/>
Show y-coordinate	<input checked="" type="checkbox"/>
Show z-coordinate	<input checked="" type="checkbox"/>
Double x-coordinate	<input type="checkbox"/>
Double y-coordinate	<input type="checkbox"/>
Double z-coordinate	<input type="checkbox"/>
Absolute x-coordinate	<input type="checkbox"/>
Absolute y-coordinate	<input type="checkbox"/>
Absolute z-coordinate	<input type="checkbox"/>
Dimensioning	
Show unit	<input checked="" type="checkbox"/>
Floating format precision	2
Unit	Millimeter ▼
Text	
Font	Arial ▼
Font size	14
Color	<input type="color" value="#FF0000"/>
Display	
Allow scaling	<input type="checkbox"/>
Align to screen	<input checked="" type="checkbox"/>
Always on top	<input checked="" type="checkbox"/>
Alignment	None ▼
Frame	
Visible	<input checked="" type="checkbox"/>
Color	<input type="color" value="#FF0000"/>
Type	Rectangle ▼
Background	
Visible	<input checked="" type="checkbox"/>
Color	<input type="color" value="#FFFFFF"/>
Line	
Color	<input type="color" value="#FF0000"/>
Connection type	Direct ▼
Endtype	Arrow ▼
Cropped	<input type="checkbox"/>

1.13 Dimensioning

In **DIMENSIONING** mode, the results of a measurement can be created as an annotation (markup) in the model area. The dimensioning annotations are generated in the structure as dimensioning below the node **CREATEDNODES** > **DIMENSIONINGSET**:

- Enable the **DIMENSIONING** mode on the **MEASUREMENT** tab before you start your dimensioning and select one of the measurement functions.
- In addition to the entries we have learned about in chapter [Measuring](#), you can specify various parameters for the creation of dimensioning annotations. These settings will apply to annotations you will create in future. End the currently selected function first if you want to edit existing markups. Then you can select the desired object, delete it or edit it as usual in the **PROPERTIES** pane (see chapter [View and Edit Properties of Selected Objects](#)).
- Proceed the same way as if you were using the measurement functions from chapter [Measuring](#).
- The result will be generated as an annotation in the model area. You can then drag and drop it with the left mouse button to place it again.
- The annotation created during dimensioning can be stored in a view. We also recommend creating a view before you start dimensioning, so that you can return to the status before you applied any dimensioning at any time (see chapter [Views](#)).

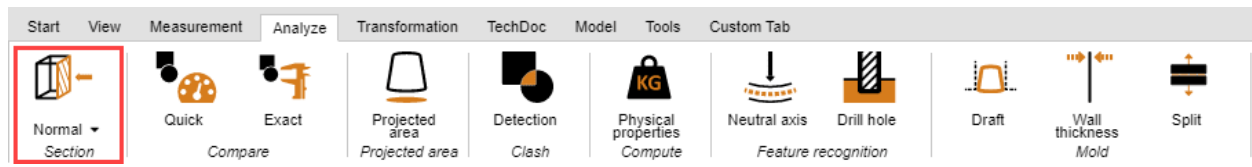


1.14 Section

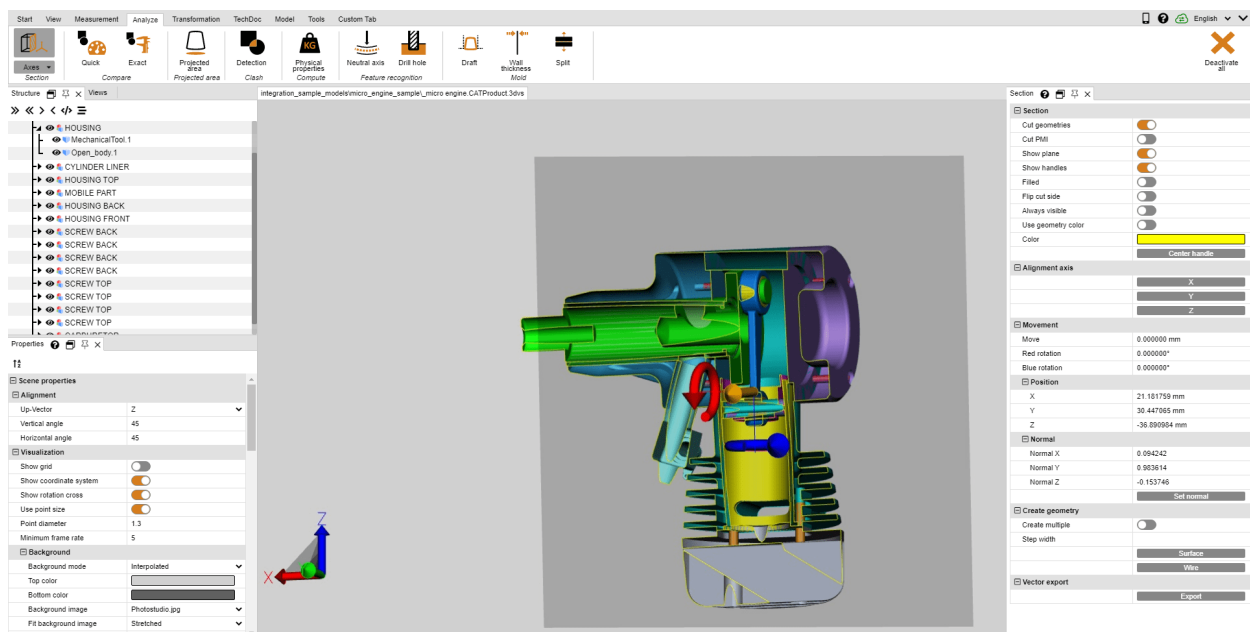
You can furthermore place multiple types of sections across models:

Function	Description
NORMAL	Generates a cutting plane at a selected point above the normal of the surface.
AXES	Generates a cutting plane at the center of the model's Bounding Box.
3 POINTS	Generates a cutting plane across any three specified points.
3 CIRCLE CENTERS	Generates a cutting plane across the centers of three selected circles.

1. Activate the section function by clicking the corresponding function button in the **ANALYZE** tab.



2. The tool pane **SECTION** appears, where the cutting plane properties can be modified.
3. Move the cutting plane by clicking the auxiliary objects and move the cursor or enter relevant values in the **MOVEMENT** area.
MOVE and **ROTATE** return the offset to the original position. In other words: entering the value "0" will return the cutting plane to its original position. The original position will be redefined by clicking the handles or by entering position coordinates.
4. Clicking the button **CENTER HANDLE** will reposition the handles centrally in the visible area if they are no longer visible (e.g. due to zooming in closely).
5. Various visualization options are provided at the top of the tool pane.
6. The currently active temporary cut can be exported as a vector graph as needed (DXF and SVG). Using the **SCREENSHOT** function in the **TOOLS** tab, the entire model area including the section can be saved.
7. The cut can also be created as a geometry in the structure of the model, making it available for further analysis and export. The cut geometry can be generated as a wire or an area.
With a **STEP WIDTH** of "0.0" or if **CREATE MULTIPLE** is disabled, only the current section is created as geometry. Multiple steps can be generated if a step width value is entered in the model unit. These can be found in the model structure in the newly created node **CREATEDNODES**.



8. Exit the section function by clicking the button again or by pressing [ESC].

1.15 Comparison of Parts, Assemblies and Drawings (3D and 2D)

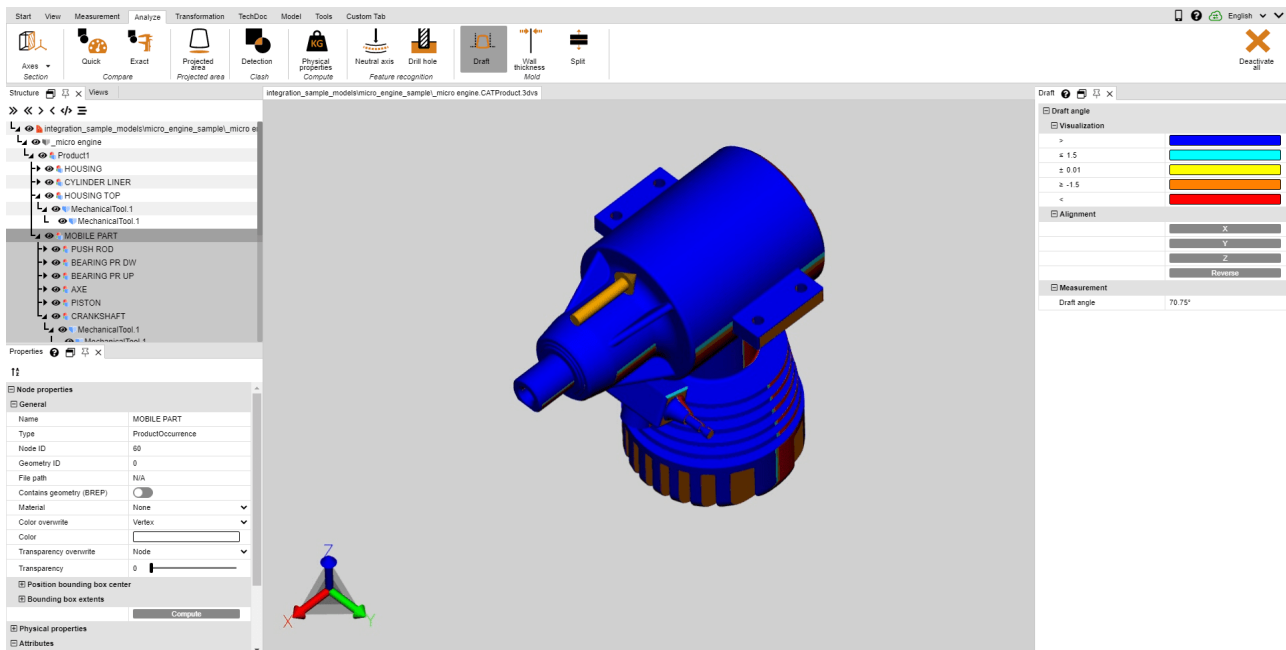
The comparison function allows you to find differences between individual parts, entire assemblies, or drawings. Here you can choose between quick compare (3D and 2D) and exact compare (only 3D).

Quick compare is based on the displayed 3D model scene or the 2D drawing without any prior calculation and offers a quick overview using coloring of the visible surfaces.

An exact compare, on the other hand, is possible only on the basis of 3D geometry data (BREP) and requires a calculation of the result. It is anchored in the model structure as a new geometry under the node **COMPARERESULTS** and will therefore be available for further analysis and export. The unchanged geometries and those that are identical in both models, as well as the geometries of the first model (Name-of-the-first-Model) that differ from those of the second model, and the geometries of the second model (Name-of-the-second-Model) that differ from those of the first model.

1.16 Draft

In the tab **ANALYZE**, you will find the tool **DRAFT** for the evaluation of possible demolding directions of a part or assembly. Depending on the specified direction, the entire visible model will be colored according to specification.




Proceed as follows to initiate a draft analysis:

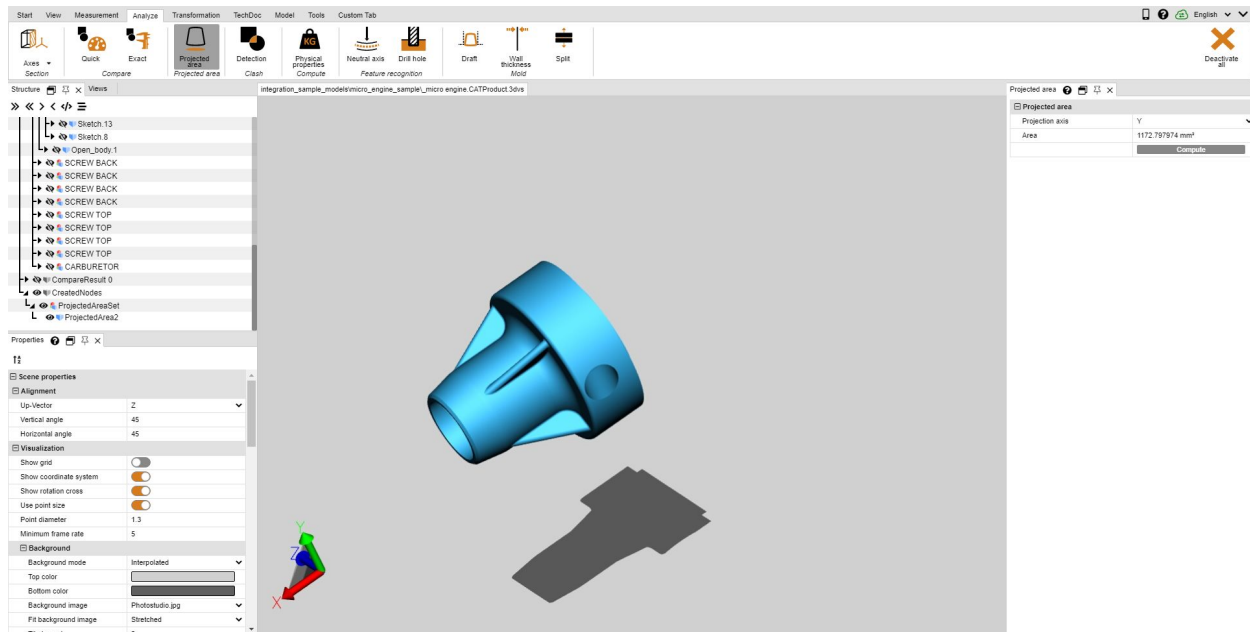
- Start the draft analysis; the cursor is displayed as an orange arrow.
- Move the cursor over the model. The cursor will always be perpendicular to the area below. It will also follow line objects and coordinate system axes to verify specified directions in the model as needed.
- Click the left mouse button at the desired positions to color the model in accordance with that draft direction.
- The draft angles and their associated colors can be specified in the tool pane. Areas positioned at the specified angle to the draft direction will be colored accordingly.
- The tool pane additionally offers the option to reverse the draft direction or to align it with the current coordinate system (chapter [Generate Coordinate System](#) 55).
- The orange cursor for aligning the demolding direction can also be aligned with lines and coordinate systems.
- Move the cursor over the model again to display the angle of the area perpendicular under the cursor in relation to the current draft direction in the tool pane under **DRAFT**.

1.17 Projected Area

In the **ANALYZE** functional area, you can use the **PROJECTED AREA** tool to visualize the projected area for parts and assemblies and compute their surface area.

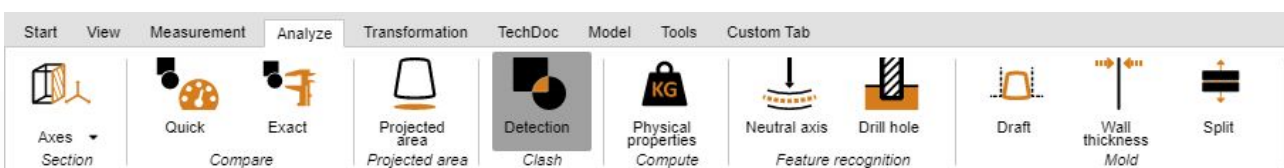
Proceed as follows to calculate the projected area of a selection:


- Generate a selection.
- In the tool area, specify the projection axis relative to the current coordinate system (see chapter [Generate Coordinate System](#) )
- Click the **COMPUTE** button.
- The projected area is now highlighted in orange.
- The surface area is displayed in the **TOOL** area under the **SURFACE** section.



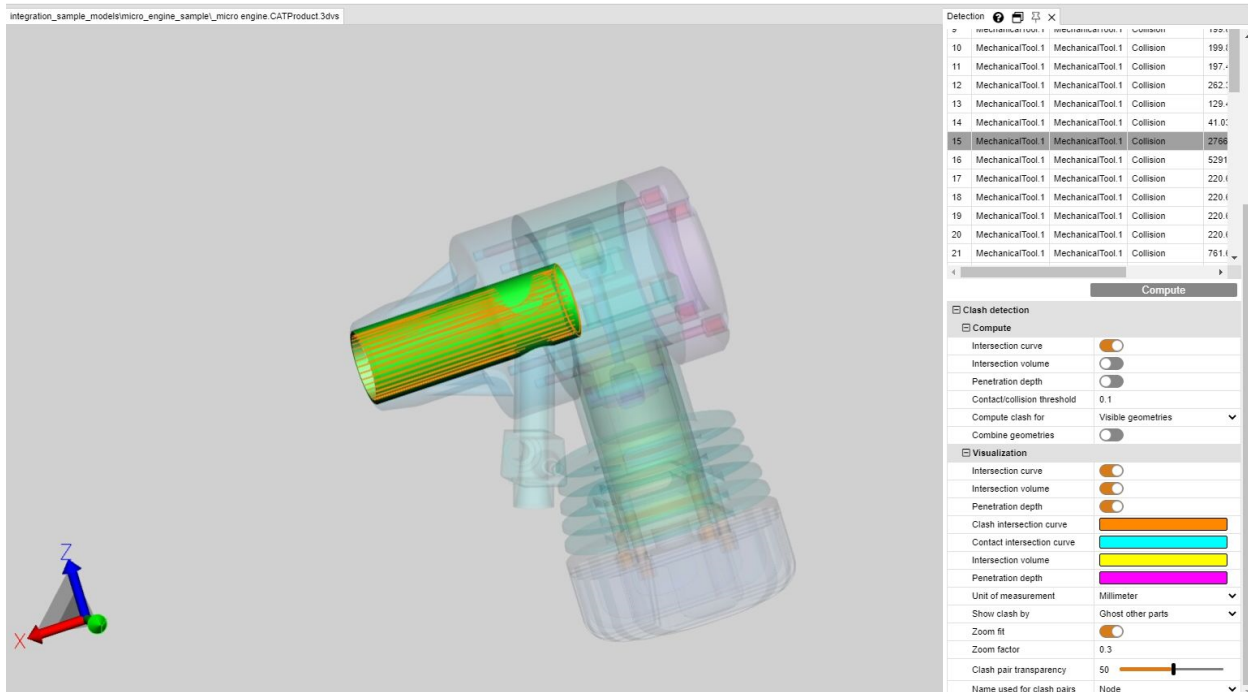
1.18 Clash Detection

The Clash Detection calculates a list of the clashing parts in an assembly. You can also compute additional parameters like **PENETRATION DEPTH** and select various types of visualizations.



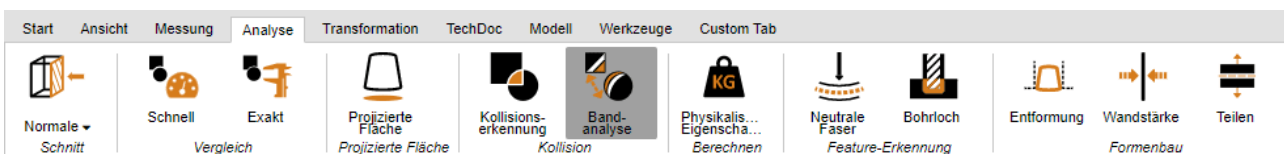
- Activate the **CLASH DETECTION** function by clicking the button with the same name in the **ANALYZE** tab.
- You can select the following parameters for clash detection in the Tools pane:
 - a. Selecting the checkbox **COMPUTE DETAILS** will compute the type of contact **CONTACT/COLLISION** and any penetration depth for the collision pair list **NODE NAMES - NODE NAME**. In addition the included points and edges are calculated so that they can be displayed directly for visualization (see bullet point 4.).
 - b. **CONTACT/COLLISION THRESHOLD** specifies the penetration depth in the model unit, from which collision pairs are no longer classified as **CONTACT**. This tolerance value can be used to e.g. balance out any inaccuracies in the tessellation.
 - c. You can select the parts for which a possible clash is to be examined from the **COMPUTE CLASH FOR** selection list:
 - **VISIBLE PARTS**: All parts will be included that are not ghosted or hidden.
 - **SAVED SELECTION SETS**: Computes collisions only for previously [saved selection sets](#) .
- Initiate the clash detection via the **COMPUTE** button. The computing time will depend on the model size and processing power of your computer. You can continue interacting with 3DViewStation while the process is running. The progress of a current computation of detail information can be viewed via the button **[COMPUTING]**.

- By selecting a row in the clash list, you can highlight the corresponding pair and specify the type of display in the section **VISUALIZATION**:
 - Show clash by:
 - Selection: The pair is selected.
 - Hide other parts: All parts except the selected pair are hidden.
 - Ghost other parts: All parts except the selected pair are displayed as in ghost mode.
- a. Zoom fit: The selected collision pair is zoomed to fit the image.
- b. Clash pair transparency: Transparency control for the analysis of clashes in covered areas.



1.19 Distance and band analysis

In the **ANALYSIS** functional area, you can use the **BAND ANALYSIS** tool to graphically display the distance between geometries (surfaces and solids).



The **BAND ANALYSIS** tool window is divided into three sections:

- **MINIMAL DISTANCE**,
- **BAND ANALYSIS** and
- **DIMENSIONING**.

Minimum Distance & Dimensioning

Analogous to the **DISTANCE: SOLID TO SOLID** function in the **MEASUREMENT** tab, the smallest distance between two selected geometries (surfaces and solids) can be dimensioned here. This is useful, for example, to define the absolute color bands.

Settings for the annotations (3D markups) of the minimum distance can be found in the **DIMENSIONING** section. If you only want to get the measurement value without 3D annotation, you can switch from **DIMENSIONING** to **MEASUREMENT** in the **MEASUREMENT** tab.

Band Analysis

The calculation of the band analysis can be absolute or relative (see below).

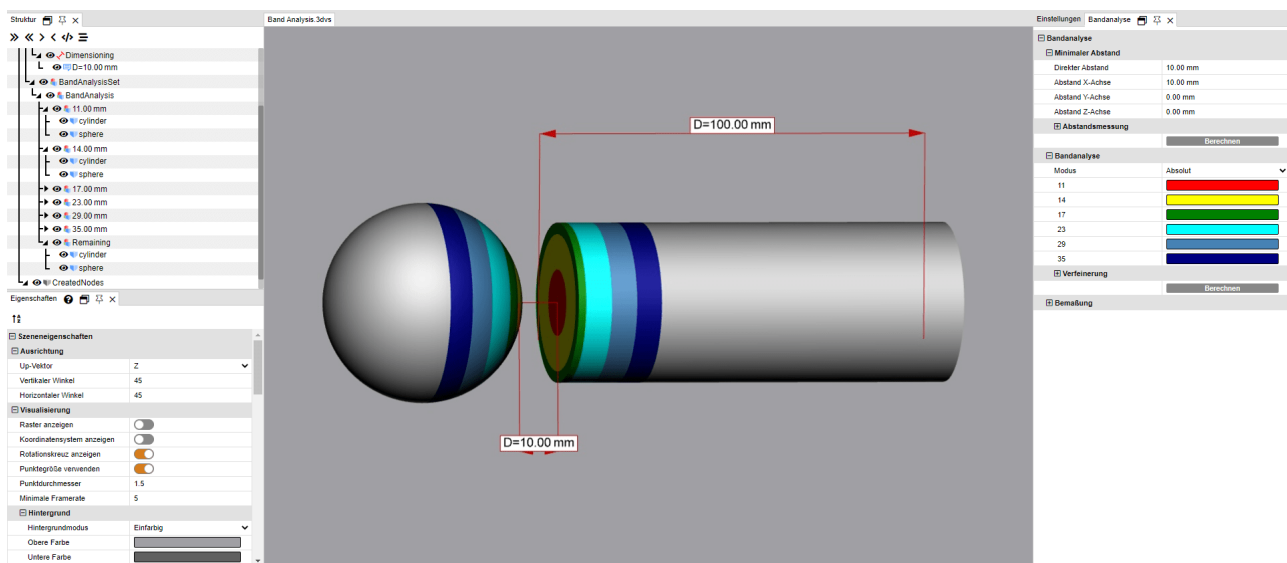
You can define up to 6 bands for the distance visualization. If you enter 0, this band is not calculated.

The resolution of the considered points for the visualization of the distance calculation is determined automatically, but can be defined manually under **REFINEMENT**. Smaller values lead to finer transitions of the bands, but also to a longer calculation time. The resolution must not exceed the width of the smallest band.

By the calculation of the bands the original geometries are hidden and colored geometries are created. These are stored in the model structure under **CREATEDNODES > BANDANALYSISSET > BANDANALYSIS 1** etc. Each band as well as the geometry parts that were not captured represent own geometries and can be selected. Thus further processing, dimensioning and export are possible.

Absolute Band Analysis

Takes into account the minimum distance between the selected geometries and colors the bands according to the specification. To create bands according to the following example, you must specify the absolute distance (minimum distance + width of the previous band).



Relative Band Analysis

The relative calculation, however, does not take into account the minimum distance between the geometries and the width of the previous band. Therefore, you only need to specify the desired width for the respective band.

As a result, the relative calculation can now be used for further analyses with geometries of different distances without any further adjustment.

The following information leads to the same visualization as in the above example for the absolute calculation:

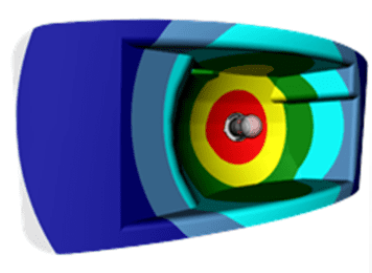
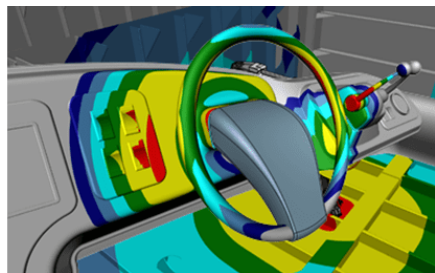
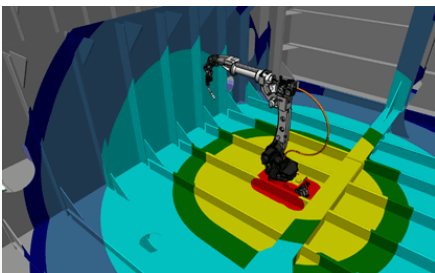
Einstellungen		Nachbarschaftssuche	Bandanalyse			
<input type="checkbox"/> Bandanalyse						
<input type="checkbox"/> Minimaler Abstand						
Direkter Abstand			10			
Abstand X-Achse			10			
Abstand Y-Achse			0			
Abstand Z-Achse			-0			
<input type="checkbox"/> Abstandsmessung						
						Berechnen
<input type="checkbox"/> Bandanalyse						
Modus			Relativ			▼
1						
3						
3						
6						
6						
6						
<input type="checkbox"/> Verfeinerung						
						Berechnen
<input type="checkbox"/> Bemaßung						

Tip



- You want to combine several geometries into one object for analysis? You will find this option for a selection of geometries in the **MODEL** tab in the **MERGE OBJECTS** tool.
- The bands are initially created without edges. If they are required for measurements or transformation, for example, you can generate them using the **GENERATE WIRES** function in the **TOOLS** tab under **GEOMETRY TOOLS**.


Examples




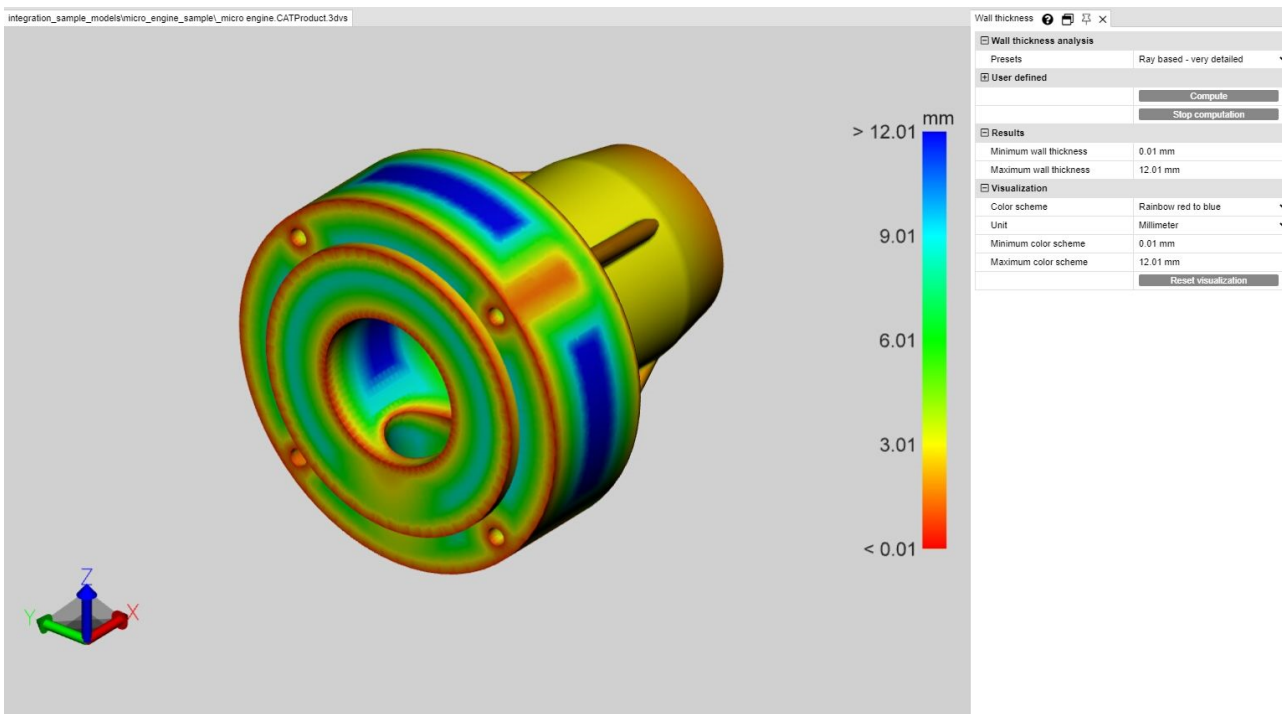
1.20 Wall Thickness: Heat Map

The **WALL THICKNESS** function: **HEATMAP** can be accessed via the **ANALYZE > HEAT MAP** tab.



In contrast to the punctual wall thickness measurements ([Measurement](#)  tab), the entire surface of a selection is colored in the wall thickness heat map analysis. The color assigned to the wall thickness can be taken from the color scale on the right edge of the model area.

Analogous to the wall thickness measurement, the wall thickness heat map can be calculated on the basis of the ray based or rolling ball (sphere) method (see [paragraph below](#) ). The default settings allow you to choose between both methods and a faster, more detailed or user-defined calculation.



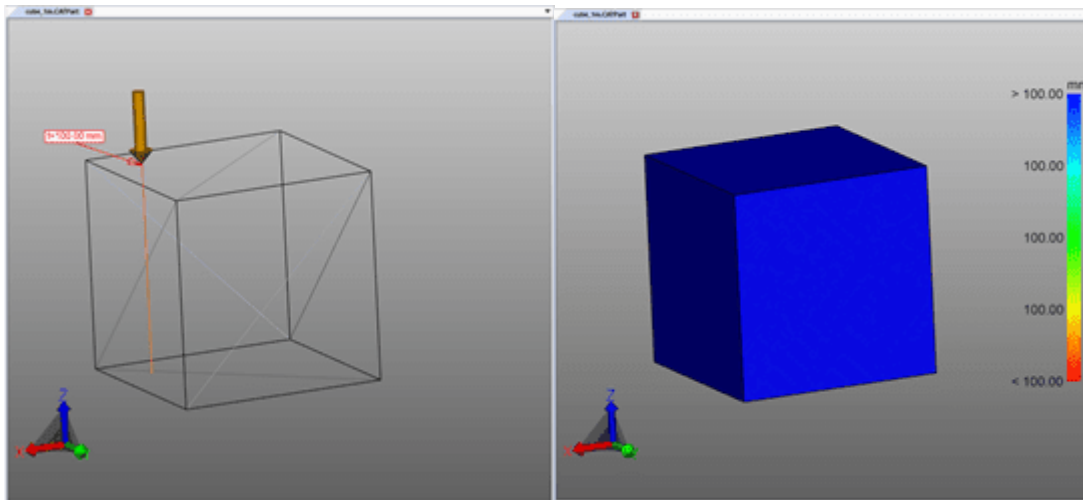
Difference between ray based and rolling ball (sphere) methods

To illustrate the difference and the value of the two methods, it is useful to display the tessellation by pressing the **[F9]** key (the left screenshot in the examples below). In general, higher tessellation qualities (more triangles) offer more detailed results .

Ray based

The following screenshot shows a punctual wall thickness measurement on the left (tab **MEASUREMENT > WALL THICKNESS > RAY**). The orange mouse pointer stands perpendicular to the underlying tessellation triangle. To determine the wall thickness, a ray perpendicular to the triangular surface was fired at this point until another triangle was hit. The two penetration points generated in this way define the distance or wall thickness measured here.

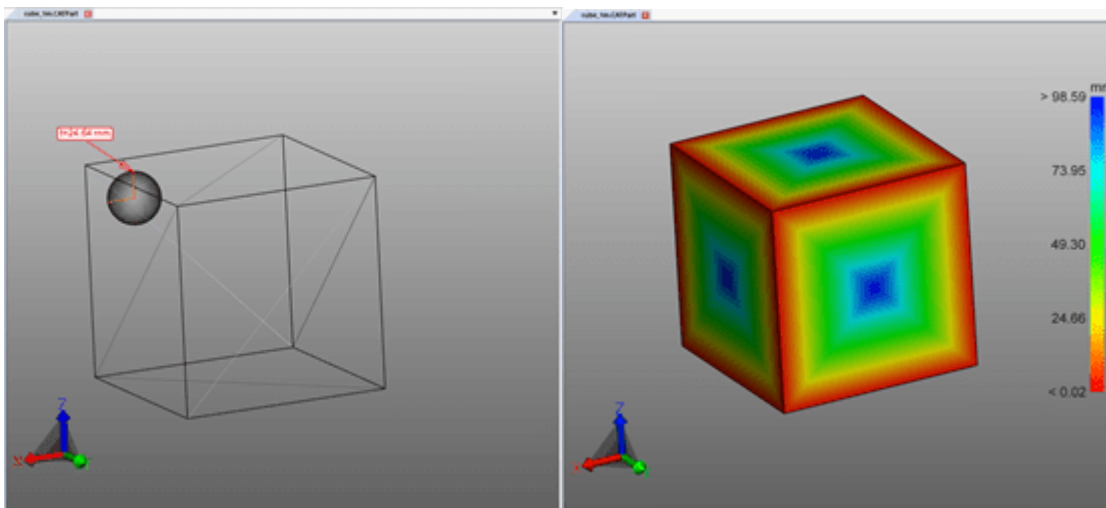
To calculate the wall thickness heat map (screenshot on the right), the number of triangles is increased depending on the setting or default setting and a ray is fired at the triangle center of gravity for each triangle. The model is colored interpolated according to the determined distances over the triangles.



Rolling ball (Sphere)

The following screenshot shows a punctual wall thickness measurement on the left (tab **MEASUREMENT** > **WALL THICKNESS** > **SPHERE**). At the clicked point, a sphere is created tangentially and extended until the sphere touches another triangle of the tessellation. The diameter of the sphere indicates the wall thickness.

To calculate the wall thickness heat map (screenshot on the right), the number of triangles is increased depending on the setting or default setting and a sphere is created at the triangle center of gravity for each triangle. The model is colored interpolated according to the determined diameter over the triangles.




Create wall thickness heat map

Proceed as follows to generate a wall thickness heat map:

- Activate the **WALL THICKNESS** function via the **ANALYZE** tab.
- Select the parts for sampling. Without selection the wall thickness is calculated for all visible parts.
- A preset can be selected or user-defined settings can be made in the tool window.
- Click the [**COMPUTE**] button. The calculation duration depends on the combination of tessellation quality, settings/presettings, selection and the available computing power.
- Once the process is complete, the original geometry disappears and the wall thickness analysis result is displayed.

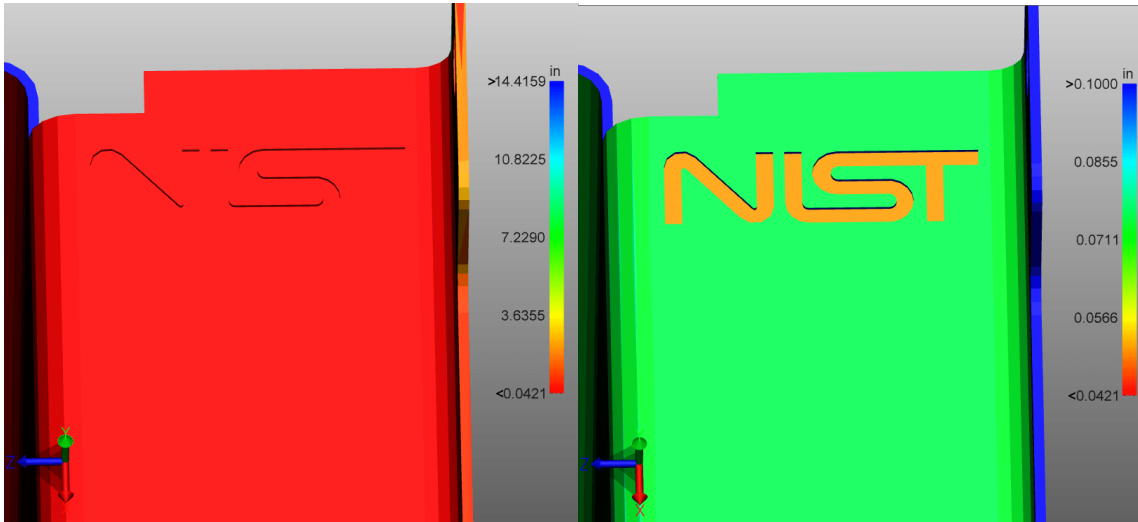
Further information:

- The wall thickness heat map is anchored as a new geometry **WALL THICKNESS CALCULATION** under the **CREATEDNODES/HEATMAPSET** node. Like other geometries, the heat map can be hidden, scaled and edited (see chapter [Selection Mode to Hide, Delete and Ghost Objects](#) )
- The color scheme of the legend is initially adjusted to the computed minimum and maximum values. They correspond to the values in the **RESULTS** section.
- For better visualization, the interval of the color scheme can be limited to the respective wall thicknesses (see figure below), in order to visualize small changes in wall thickness at certain intervals.

- In the following images you first see the result with the calculated maximum and minimum wall thickness, followed by a visualization with manually restricted maximum wall thickness, whereby the lettering becomes visible.

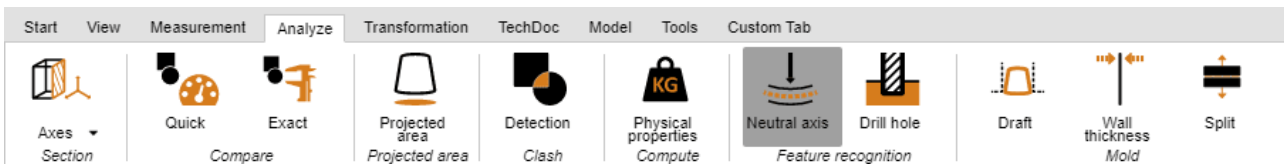
In the bottom right figure, only the maximum value of the legend is limited to 0.1 inches. Thus, all wall thicknesses greater than 0.1 inch are colored blue, but the entire color scheme is now divided into the interval of 0.0421 and 0.1 inch. As a result, the lettering becomes visible, since this small change in wall thickness is already assigned to a new color.

The heat map of the wall thickness analysis can be saved in a view as usual (see chapter [Views](#) ¹⁷).

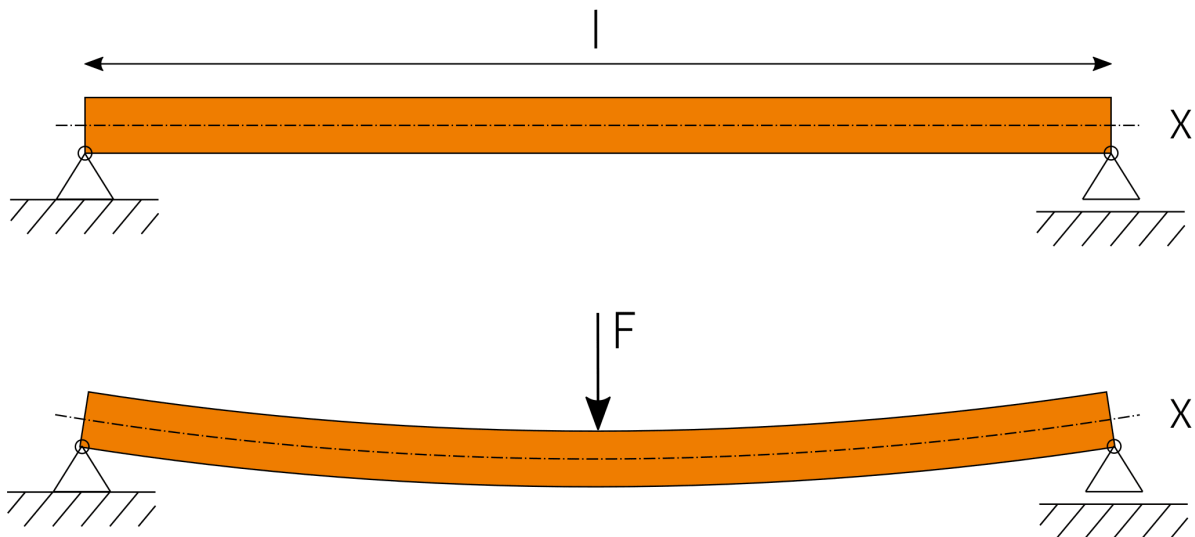
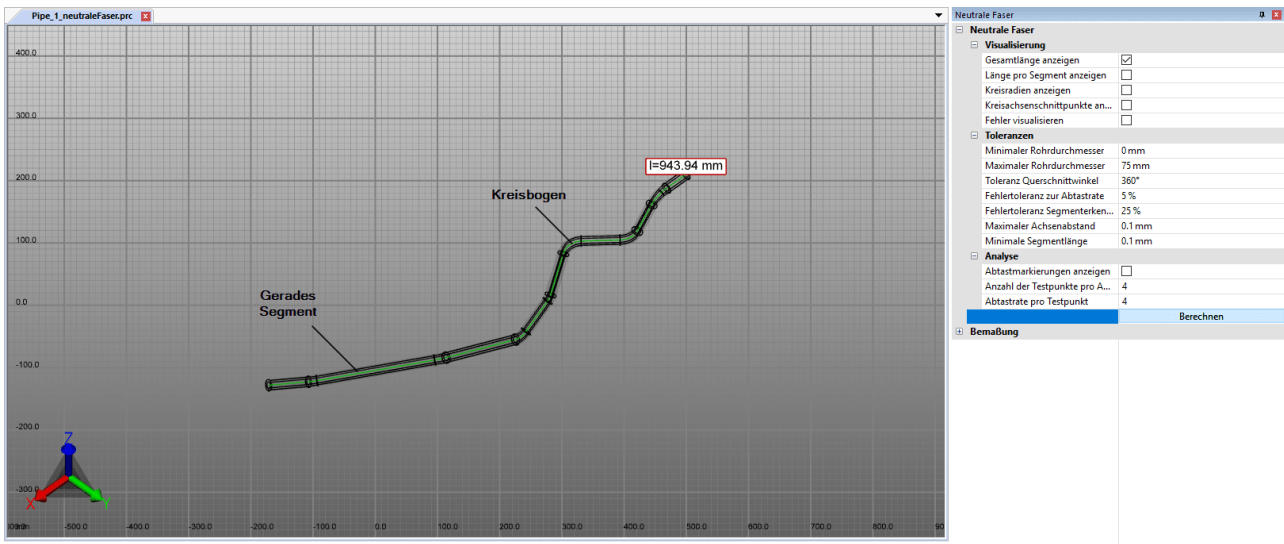


1.21 Neutral Axis

The function **NEUTRAL AXIS** can be accessed via the **ANALYZE > FEATURE RECOGNITION** tab.



A neutral axis (zero line) is the axis or layer of a beam or tube cross-section whose length does not change with twisting or bending. With this tool the stretched length for circular tubes can be determined, independent of their bending. The neutral axis is calculated on the basis of individual segments. A segment is formed by two circles connected by a straight line or forming an arc.



To calculate the neutral axis, proceed as follows:

- Activate the **NEUTRAL AXIS** function via the **ANALYZE** tab.
- Create a selection for the calculation. Without a selection, the neutral axis is calculated for all visible geometries with a circular cross-section.
- If necessary, adjust the settings and then start the analysis using the **[COMPUTE]** button.
- Geometries for visualization and dimensioning are created as **NEUTRALAXISSET** under the **CREATEDNODES** node.

To obtain information on the individual settings, click on a setting in the tool area. You will then receive detailed descriptions of the respective setting in the lower area of the windows - in the so-called Help Box.

1.22 Transformations (Move, Rotate, Mirror and Explode)

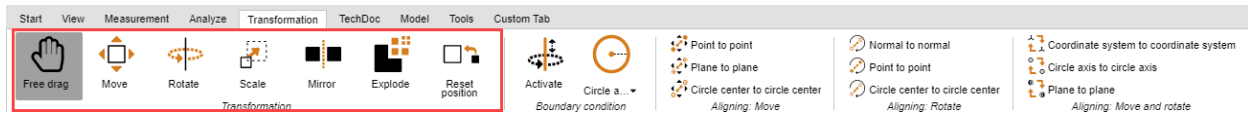
Make sure to familiarize yourself with the relevant chapters [Object Selection](#) and [Saving Selections](#), because all the transformations described in the following chapter are done for a selection.

You have several options for transforming objects:

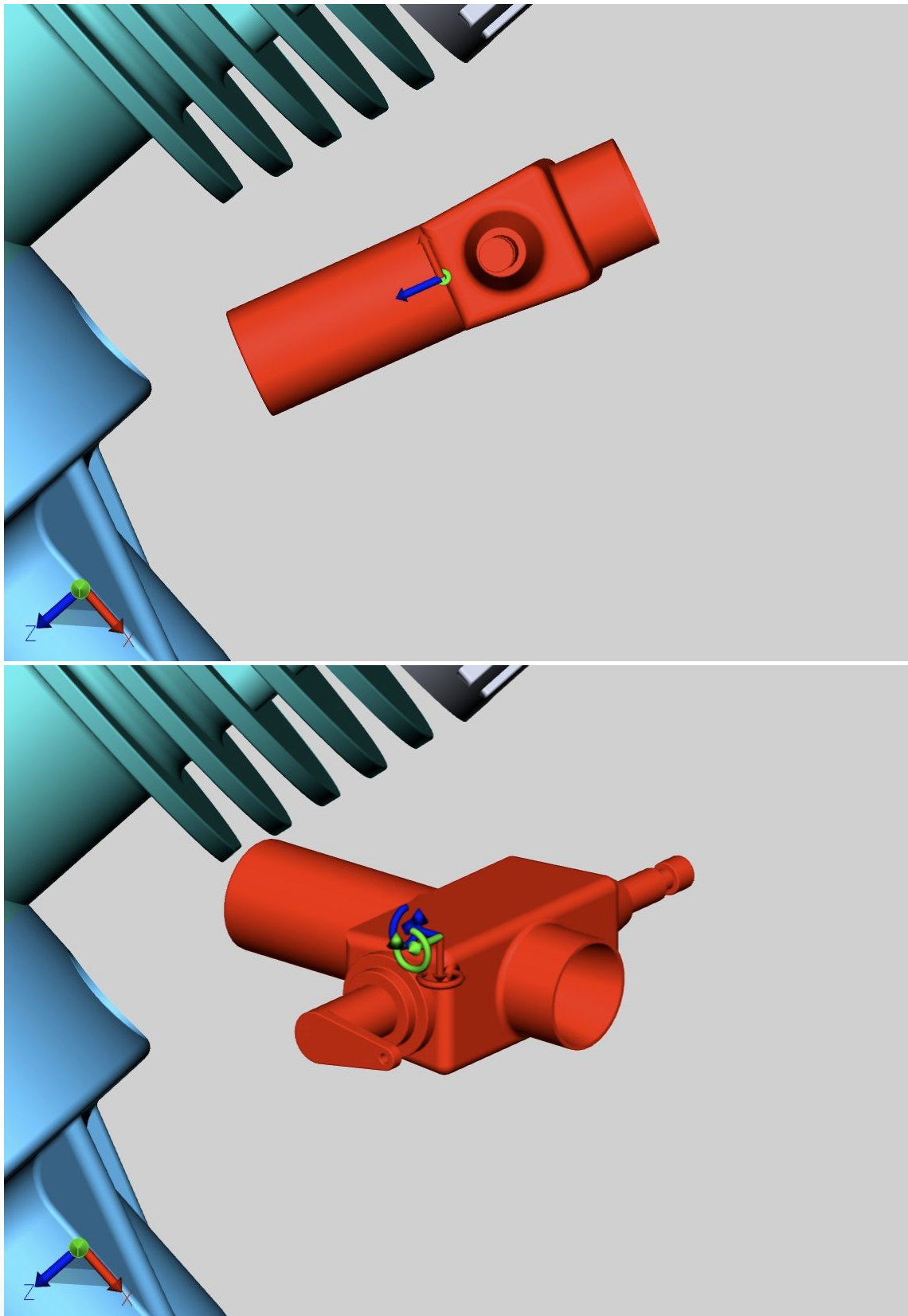
- Transformation via Auxiliary Objects (Handles)
- Transformation by way of value entry
- Transformation with boundary conditions
- Transformation via reference objects

Transformation via Auxiliary Objects (Handles)

- Select the object(s) of interest via mouse click (if necessary with multiple selection using the keys [CTRL] or [⇧]) in the model area or via the structure.
- Enable the relevant transformation function via its button in the **TRANSFORMATIONS** tab.



- Click the handles and move the cursor to move or rotate objects directly in the model area. The **FREE DRAG** function allows the repositioning of an object via drag & drop. The object is moved within the level currently selected as the model view.
- Exit the transformation function by clicking the same button again within the function area or by pressing [ESC].
- Clicking the [**RESET POSITION**] button returns the object to its original position. Each individual transformation step can be reversed by clicking **UNDO** in the **START** ribbon.



Transformation by way of value entry

Initiate **FREE DRAG**, **MOVE** or **ROTATE**, as described in the section above. The tools pane appears, where the following options are provided for the exact change in position:

Transformation via the input of position coordinates

You will find the coordinates of your current selection under **POSITION BOUNDING BOX CENTER**. If you change the coordinates here, the selection is moved to the corresponding new position.

Transformation via the input of move values

A selection can also be moved in relation to the current position. This translation is the offset to the initial position. That means on the one hand that the selection will be reset to the initial position when the value **[0]** is entered, and on the other that various values can be tried out without having to compute previous entries. The values entered, as well as the initial position will remain intact until: the selection is deselected, a transformation is executed via the handles, or a value is entered under Rotation.

Entering rotation angle values

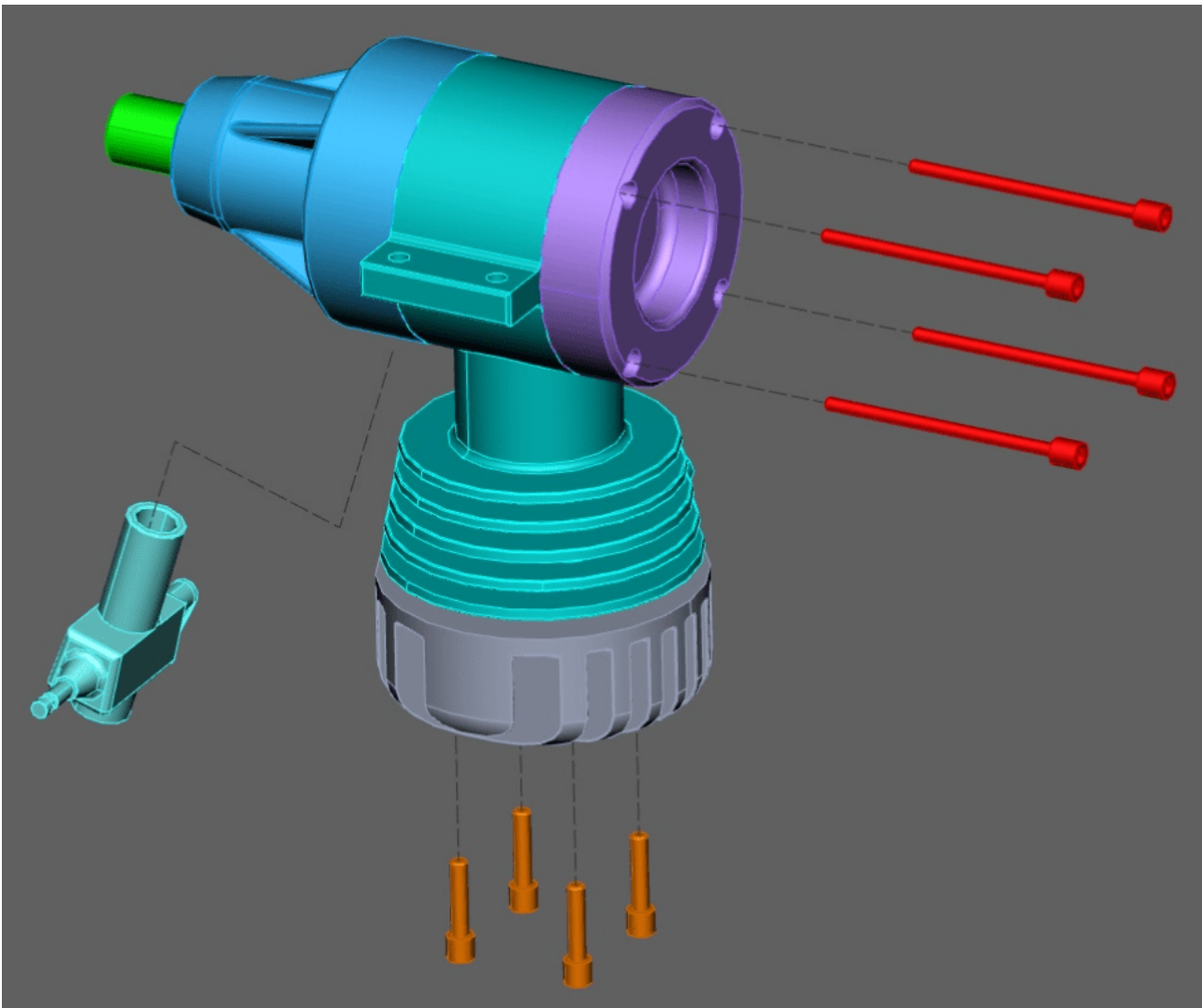
Just like for a translation, an angle value can be entered for **ROTATION** to rotate the current selection on the relevant axis. The rotation axis passes through the bounding box center, unless boundary conditions were specified.

Create offset lines

Translations can be visualized via offset lines.

- To do this, activate the **CREATE OFFSET LINE** toggle in the tool area.
- Adjust the **LINE TYPE**, **PATH TYPE** and **COLOR** if necessary.
- Use the **FREE DRAG** or **MOVE** transformation function to move the desired objects out of the assembly.
- Click the **DELETE OFFSET LINES** button in the Tool Area to remove the created offset lines.

Neutral axis	
Transform	
Position bounding box center	
X	21.50 mm
Y	28.50 mm
Z	-35.65 mm
Translation	
X	0.00 mm
Y	0.00 mm
Z	0.00 mm
Rotation	
X	0.00°
Y	0.00°
Z	0.00°
Scaling	
Scale to	100.00 %
Offset line	
Create offset line	<input checked="" type="checkbox"/>
Path style	Automated
Dash style	---
Color	

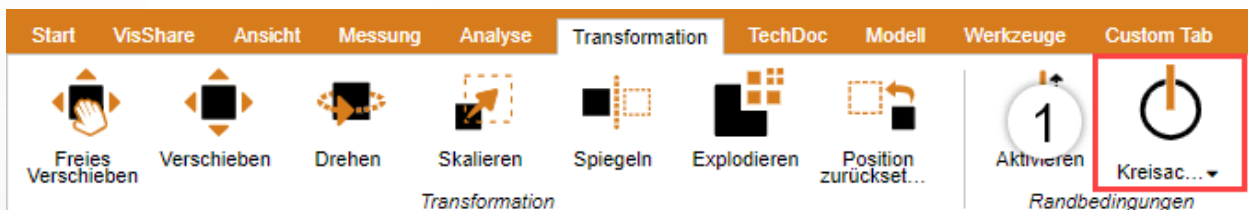


Boundary Conditions

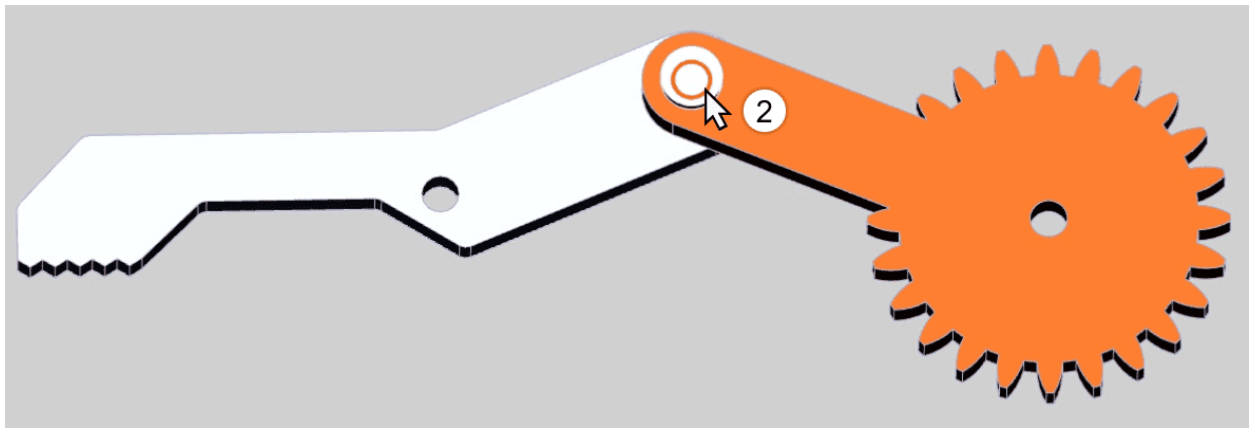
Boundary conditions allow the limitation of transformations to specified axes. This function is particularly useful for transformations via reference points. Boundary conditions can be lines, e.g. at edges of geometries, circular axes or the axes of the coordinate system.

- Choose a type for the boundary condition: **LINE**, **CIRCLE AXIS**, **X-AXIS**, **Y-AXIS** or **Z-AXIS**. In this example, the boundary condition circle axis is used.

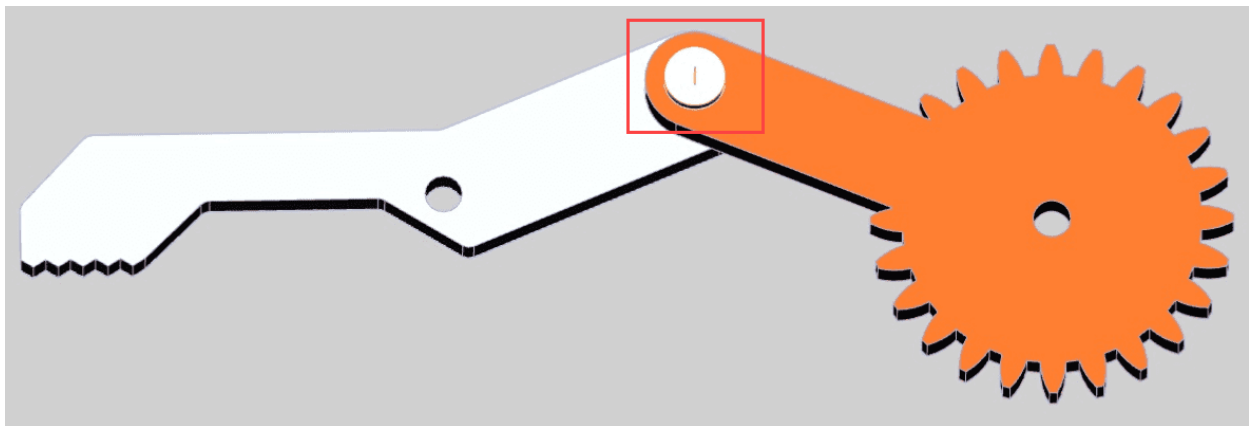
Note: If you select one of the xyz axes as boundary condition, the corresponding xyz axis of the world coordinate system is used as boundary condition. However, if you use a custom coordinate system, the corresponding xyz axis of that coordinate system will be used. See chapter [Generate Coordinate System](#)



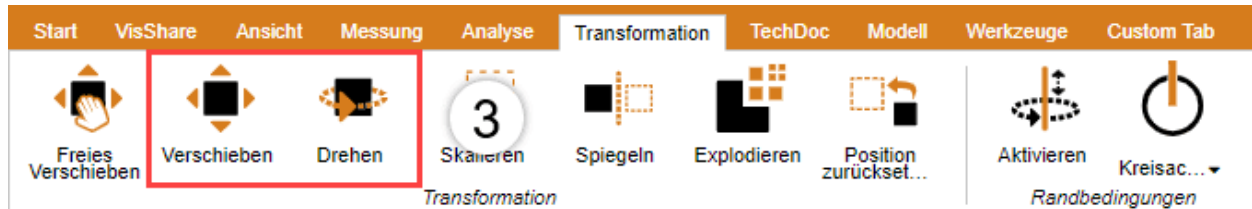
- Move the cursor over a geometry. Possible positions for boundary conditions are highlighted.



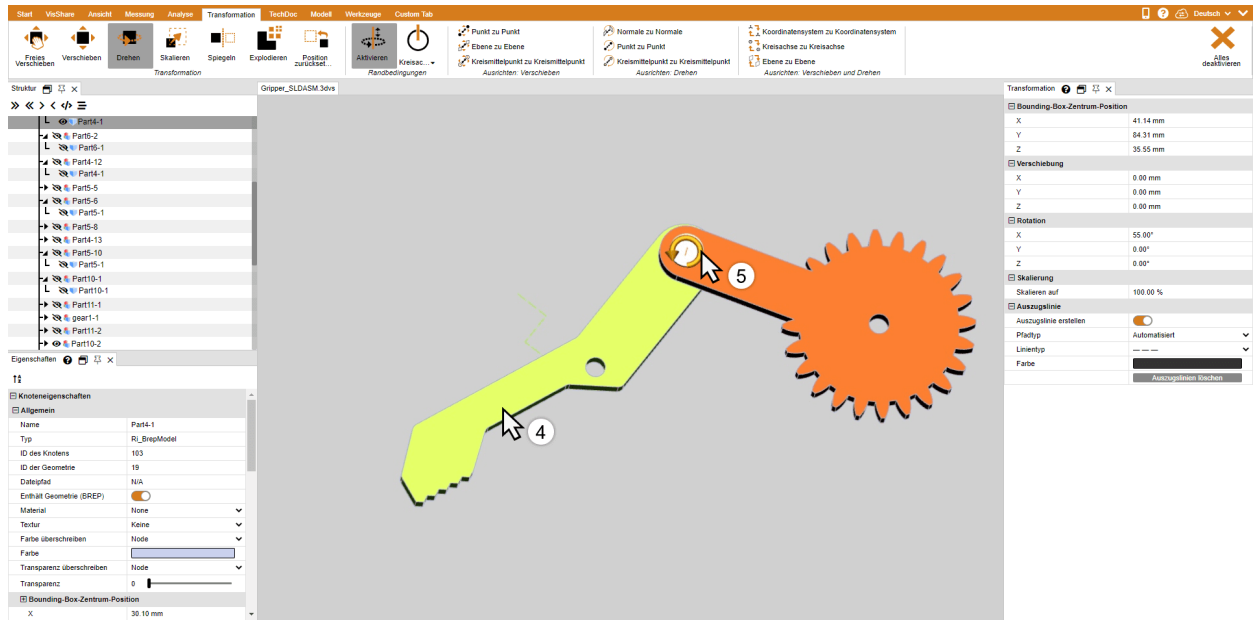
- Left-click a position to confirm.
- The selected boundary condition is activated and is visualized in the 3D scene by an orange auxiliary geometry; additionally, the button **[BOUNDARY CONDITION]** axis is now highlighted.



- Select the desired transformation mode.



- Select the object(s) you want to move or rotate around the boundary condition.
- If now, as described above, a displacement or rotation is performed, this is only possible along the boundary condition just set.



Transformation via reference points

You have several options for transforming objects via reference points. These are divided into three categories for a better overview: Move, Rotate as well as Move and Rotate. Some of these transformations will require an active boundary condition (see above).



Move

This category contains only functions for moves without rotation.

Point to Point Translation

- As a first step, select the parts you wish to move.
- Select the **POINT TO POINT** function.
- Select any starting point for the move. Lines, edges, corners, coordinate systems, etc. will be captured to place the exact initial position. Generally, you will first have to select a point on the selected object to achieve the expected transformation.
- Select the end point for the move.
- The selection will now be translated in such a way as to position the selected points congruently without any rotation.

Plane to Plane Translation

- Choose a type for the boundary condition: Line or circle axis.
- As a first step, select the parts you wish to move.
- Select the **PLANE TO PLANE** function.
- Select a plane on the part you want to translate.
- Select the target plane for the move.

Rotate

This category contains only functions for rotations without translation.

Normal to Normal

This function allows you to align objects along the normal of areas, e.g. to position two areas in parallel.

- As a first step, select the object you want to rotate.
- Select the **NORMAL TO NORMAL** function.
- Click the area for which you want to align the normal. Generally, you will first have to select an area on the selected object to achieve the expected transformation.
- Now click the area to the normal of which you want to align the normal specified in point 3.
- The selected areas will now be positioned in parallel.

Point to point

A boundary condition must be set first in order to use this transformation. You can then use this function to rotate objects based on two points around the axis defined by the boundary condition.

- Specify a boundary condition as described above.
- As a first step, select the parts you wish to move.
- In the category **ROTATE**, select the function **POINT TO POINT**.
- Select any starting point of the rotation. Lines, edges, corners, coordinate systems, etc. will be captured to place the exact initial position. Generally, you will first have to select a point on the selected object to achieve the expected transformation.
- Select the end point for the rotation.
- The object is now rotated around the previously defined boundary condition so that the points are positioned as congruently as possible or at the least possible distance from each other.

Circle Center to Circle Center

A boundary condition must be set first in order to use this transformation. You can then use this function to rotate objects based on two circle centers around the axis defined by the boundary condition.

- Specify a boundary condition as described above.
- As a first step, select the parts you wish to move.
- In the category **ROTATE**, select the function **CIRCLE CENTER TO CIRCLE CENTER**.
- Select any circle as starting point for the rotation. Generally, you will first have to select a circle on the selected object to achieve the expected transformation.
- Select a circle as the end point of the rotation.
- The object is now rotated around the previously defined boundary condition so that the circle centers are positioned as congruently as possible or at the least possible distance from each other.

Move and Rotate

This category contains functions that allow a move and rotation simultaneously.

Coordinate System to Coordinate System

You should familiarize yourself with the [creation of coordinate systems](#) before you use the function for the transformation from coordinate system to coordinate system.

- As a first step, select the parts you wish to align. The initial coordinate system can be part of that selection.
- Select the "Coordinate system to coordinate system" function.
- Select the initial coordinate system for the translation in the 3D scene (usually part of the selection).
- Select the target coordinate system for the translation in the 3D scene.
- Your selection will be translated from the initial coordinate system to the target coordinate system in such a way that the two coordinate systems are congruent.

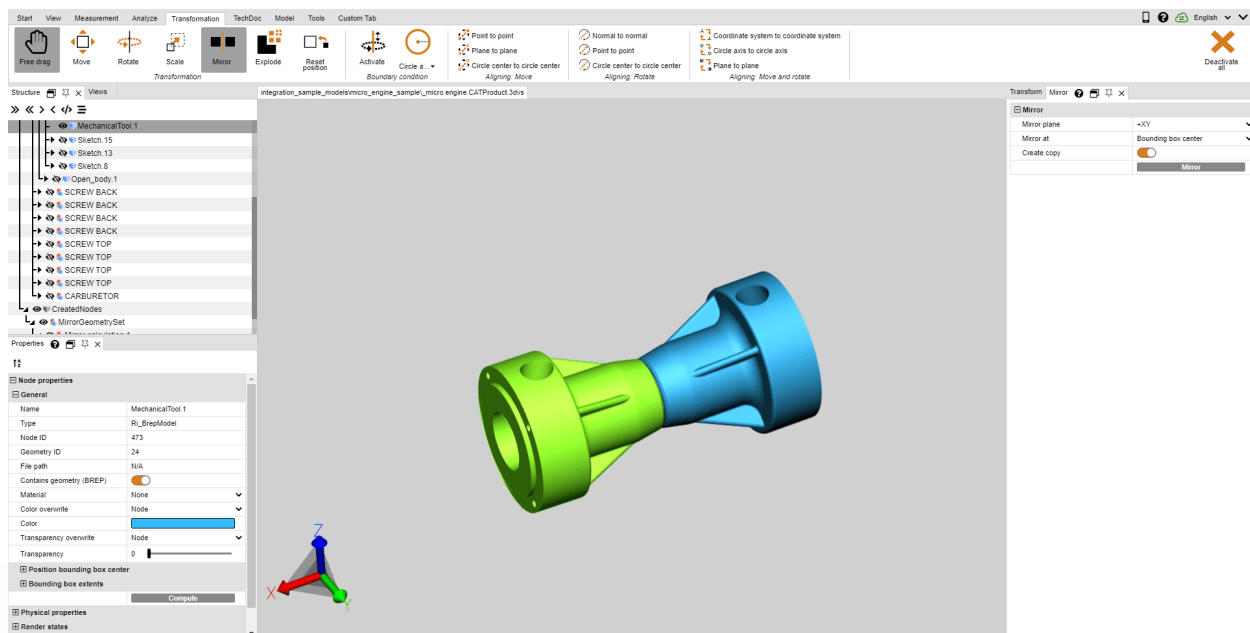
Circle Axis to Circle Axis

This function allows a positioning of your selection via circles.

1. As a first step, select the parts you wish to align.
2. Select the **CIRCLE AXIS TO CIRCLE AXIS** function.
3. Select any circle as starting point of the transformation. Generally, you will first have to select a circle on the selected object to achieve the expected transformation.
A successfully defined initial circle will be highlighted in orange. This circle will also contain a drawn coordinate system. Take note of the position of this coordinate system in relation to the circle axis for step 5.
4. Select another circle to which the previously selected circle is to be translated. Once the target circle is specified, it too will be highlighted in orange and it too will contain a drawn coordinate system.
5. You now have to specify the alignment to complete the transformation. Move the cursor to one or the other side of the target circle. The coordinate system preview on the circle axis will follow the cursor. Left-click to specify the alignment and complete the transformation.
6. The selection will now be transformed in such a way that the circles are positioned congruently, taking into account the orientation of the coordinate systems.

Reflection

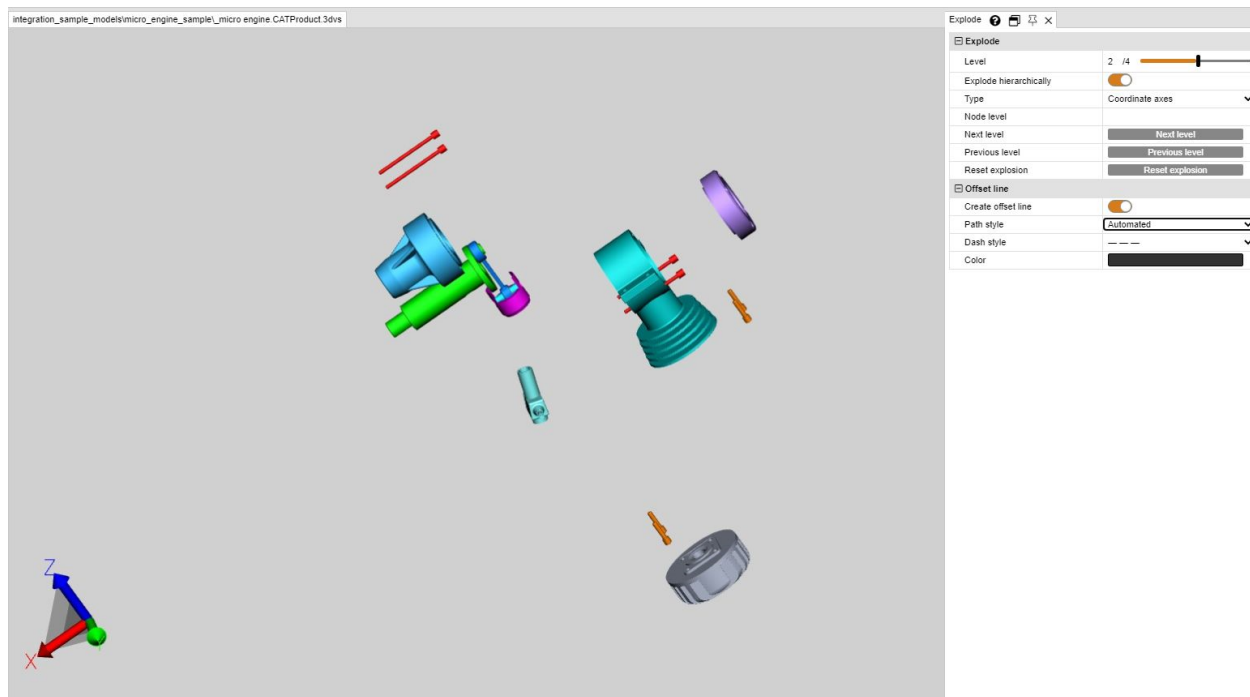
1. Select the object to be mirrored.
2. Enable the **MIRROR** function via the button in the **TRANSFORMATIONS** tab.
3. Select the mirror plane in the tool pane menu.
4. Mark the checkbox **CREATE COPY** to generate a new mirrored object instead of mirroring the original object. Objects created via the Mirror function will be saved under the structure node CreatedNodes/MirrorGeometrySet.
5. Click the **MIRROR** button in the tool pane. In the illustration below, the object was subsequently moved.



Explode

The function **EXPLODE** allows you to move the individual parts of the model from the center to the outside. You can specify whether the hierarchy planes of the structure should be included.

1. Enable the **EXPLODE** function via the button in the **TRANSFORMATIONS** tab.
2. If you want to include the hierarchy planes of the structure when exploding, mark the **EXPLODE HIERARCHICALLY** option and activate, if necessary, the **OFFSET LINES** function.
3. Use the slider to set the desired explosion level or enter it manually. The result is displayed immediately.

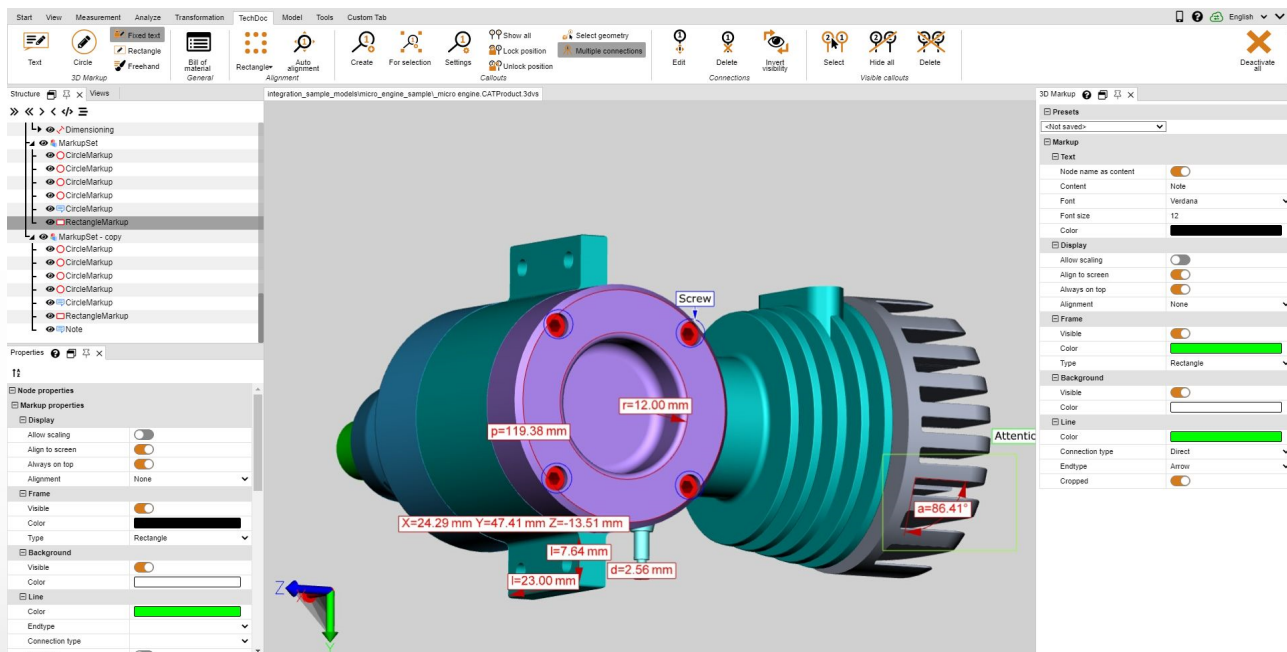


Reset Position

Any transformed parts can be reset to their original position. Select one or more parts and click the button **[RESET POSITION]**.

1.23 Annotations / Markups

The following 3D markup tools are available to attach annotations in the model area: **TEXT**, **CIRCLE**, **RECTANGLE** and **FREEHAND**. These can be stored as a view .



The following steps are required to add annotations in the model area:

- Enable the desired function in the **3D MARKUP** area of the **TOOLS** tab.
- Attributes like color can be configured in the **MARKUP** tool pane before generating the 3D markups. End the currently selected function first if you want to edit existing markups. Then you can select, delete or edit the desired object as usual in the properties pane.

- Left click the model area to specify the start point of the 3D markup, hold the button and drag the mouse to the desired end point.
- Circles are drawn from their center point.
- Text markups must be generated on a geometry.
- The text container of a text markup can be selected and moved at any time. Please use the [transformation functions](#) (32) to move an entire markup as well as circles, rectangles and free hand drawings.
- Save a view to store the 3D markups and the view of the model associated with these markups. We recommend creating a view before you start annotating, so that you can return to the status before you applied any markups at any time (see chapter [Customizing the View](#) (18)).

1.24 Filter (Search)

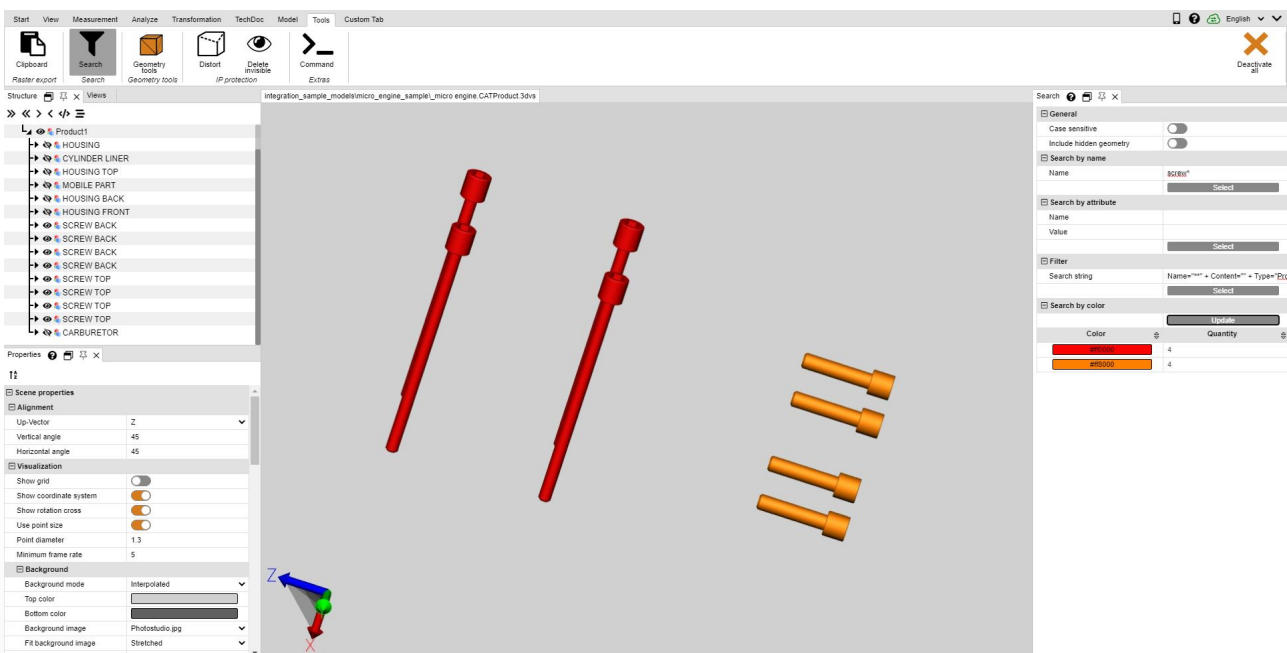
Proceed as follows to use the filter function to find objects in the model:

1. Activate the **FILTER** function in the **TOOLS** tab.
 - **SEARCH BY NAME** filters for matches with the names of all objects in the structure.
 - **SEARCH BY ATTRIBUTE** filters all attributes of all objects. Use only one of the fields or fill out both fields to limit the search results further.
 - **SEARCH BY TYPE** selects all objects of the same type.
 - **SEARCH BY COLOR** selects objects via their color property.
2. Enter a text you want to search for and start the search by pressing the **[ENTER]** button or the **[SELECT]** button.
 - You can use the "*" placeholder for any number of characters and "?" for exactly one character.

To search for objects using their color, first click the **UPDATE** button. A list of colors is displayed for all visible objects. Click on the color to be selected. The number displayed next to the color represents the number of objects of that color.

Multiple selection of colors is possible with **[CTRL]** or **[⇧]**.

3. The objects found are selected.
4. To display only the found objects in the model area, you can either right click in the model area and select the **ISOLATE** function or use **[ISOLATE]** button on the **START > SELECTION** tab.
5. Close the **FILTER** pane when you no longer need it.

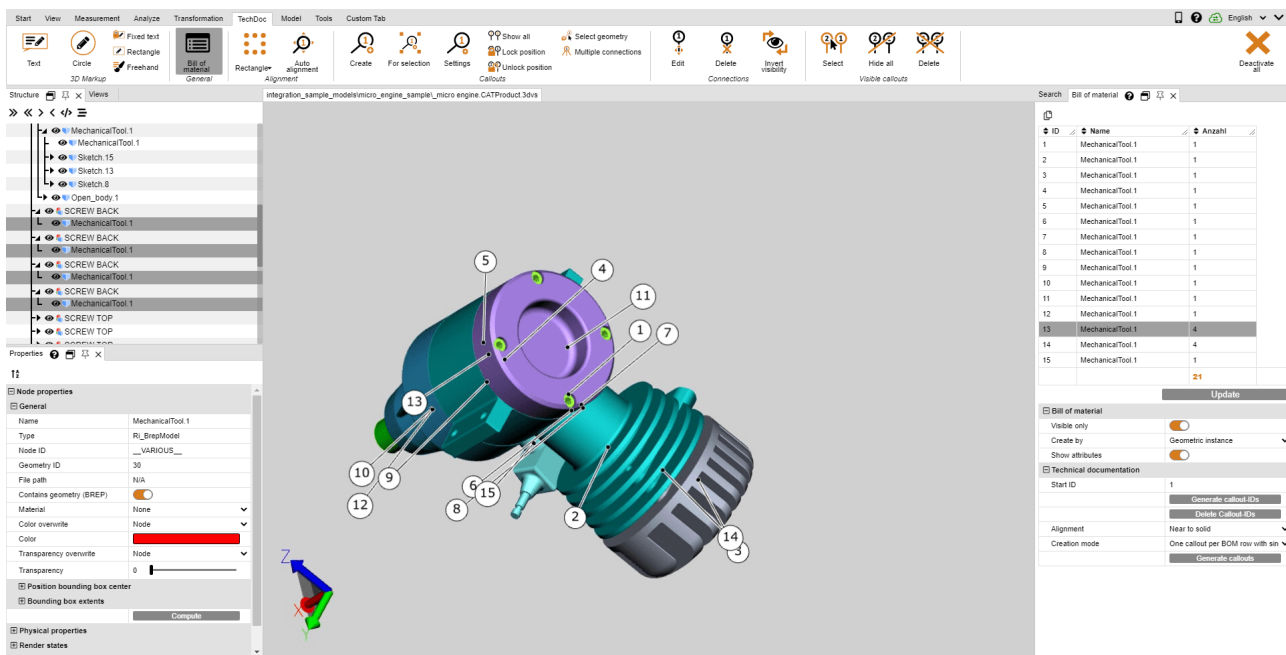


1.25 Technical Documentation (TechDoc) and Bill of Material (BOM)

With this function, you can add [Callouts](#) (49) to your model and generate technical documentation from them. The technical documentation can be generated manually or automatically from the [BOM](#) (45). You can edit callouts visually and in terms of content using numerous options.

The bill of material can be opened and used without an additional license option via the **TOOLS** tab.

To use the functions in the **TECHDOC** tab, you need a corresponding license option. If you are interested, please contact our sales department.



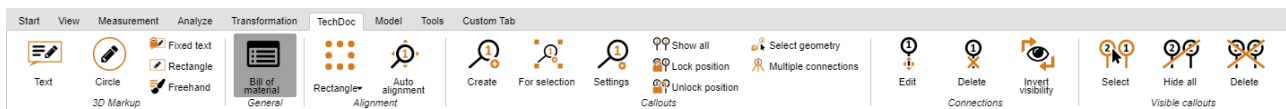
In the next chapters you will learn more about:

- [Bill of material](#) ⁴⁵
- [Callouts](#) ⁴⁹

1.25.1 Bill of material

A bill of material is a list of parts. It can be created for the entire loaded model or only for visible parts.





To do this, activate the **BOM** function on the **TOOLS** tab.




The **BOM** tool window opens in the left structure area of the 3DViewStation as a subtab.

Update	
<input type="checkbox"/> Bill of material	
Visible only	<input checked="" type="checkbox"/>
Create by	Geometric instance ▼
Show attributes	<input checked="" type="checkbox"/>
<input type="checkbox"/> Technical documentation	
Start ID	1
	<input type="button" value="Generate callout-IDs"/>
	<input type="button" value="Delete Callout-IDs"/>
Alignment	Near to solid ▼
Creation mode	One callout per BOM row with sin ▼
	<input type="button" value="Generate callouts"/>

- Use the **[UPDATE]** button to generate or update the bill of material on the basis of the stored settings and your selection in the model.


Bill of material    





ID	Name	Anzahl
1	MechanicalTool.1	1
2	MechanicalTool.1	1
3	MechanicalTool.1	1
4	MechanicalTool.1	1
5	MechanicalTool.1	1
6	MechanicalTool.1	1
7	MechanicalTool.1	1
8	MechanicalTool.1	1
9	MechanicalTool.1	1
10	MechanicalTool.1	1
11	MechanicalTool.1	1
12	MechanicalTool.1	1
13	MechanicalTool.1	4
14	MechanicalTool.1	4
15	MechanicalTool.1	1
		21



Update

Bill of material

Visible only	<input checked="" type="checkbox"/>
Create by	Geometric instance 
Show attributes	<input checked="" type="checkbox"/>

Technical documentation

Start ID	1
	Generate callout-IDs
	Delete Callout-IDs
Alignment	Near to solid 
Creation mode	One callout per BOM row with sin 
	Generate callouts

The referenced geometries can be selected via the bill of material. Accordingly, all parts selected by you in the model are highlighted in the bill of material. You can add attributes to the bill of material using the **SHOW ATTRIBUTES** setting. The small buttons below the title bar of the BOM window can be used to export a created bill of material to the clipboard () or to a CSV file (.

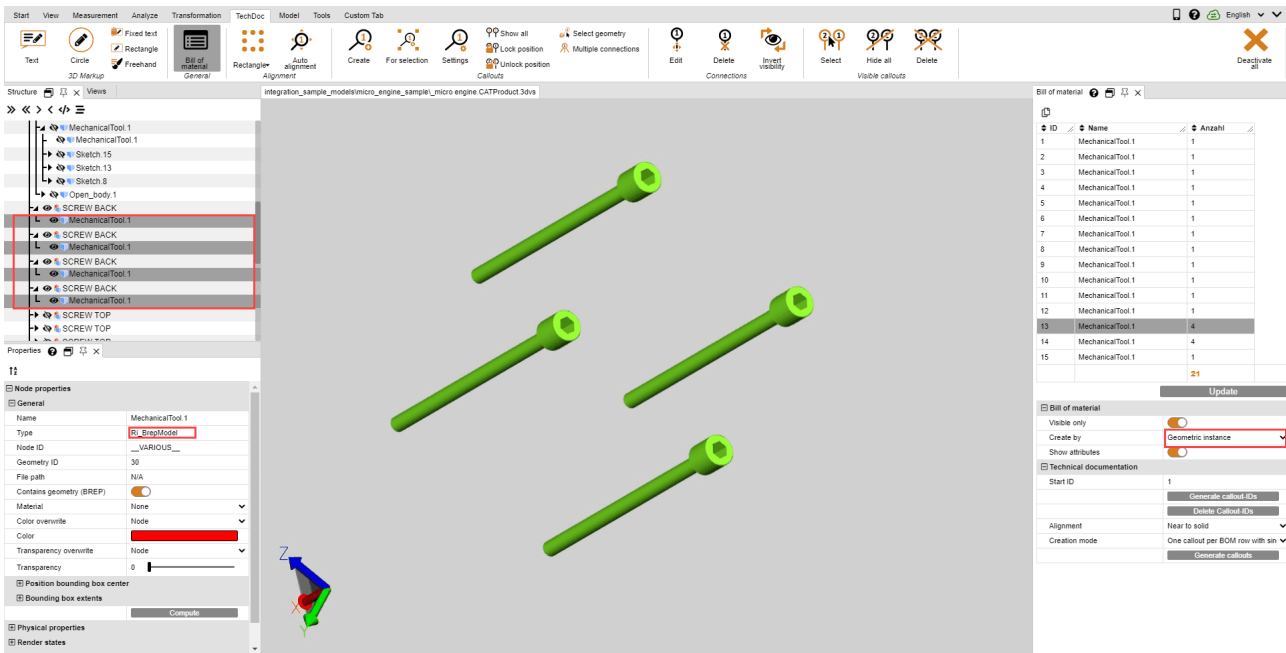
The bill of material can be created using the references described below.

Note the following:

- For a better overview, the structure window was attached to the right edge of the screen.
- In order to display the respective properties, the counted layers were selected for the images in this chapter. This is not necessary for the creation of the BOM about geometric instance and product occurrence.

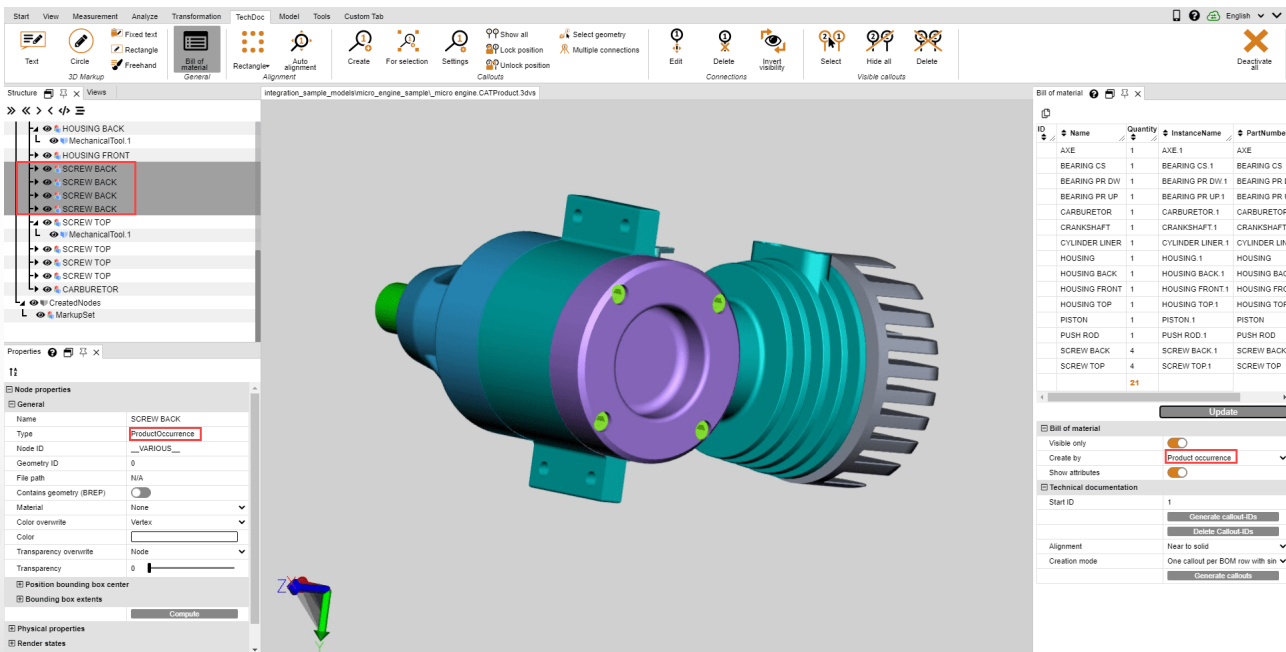
Geometric instance

This bill of material contains the number of all geometric instances (lowest structure level, e.g. type **BREPMODEL**).




Product occurrence

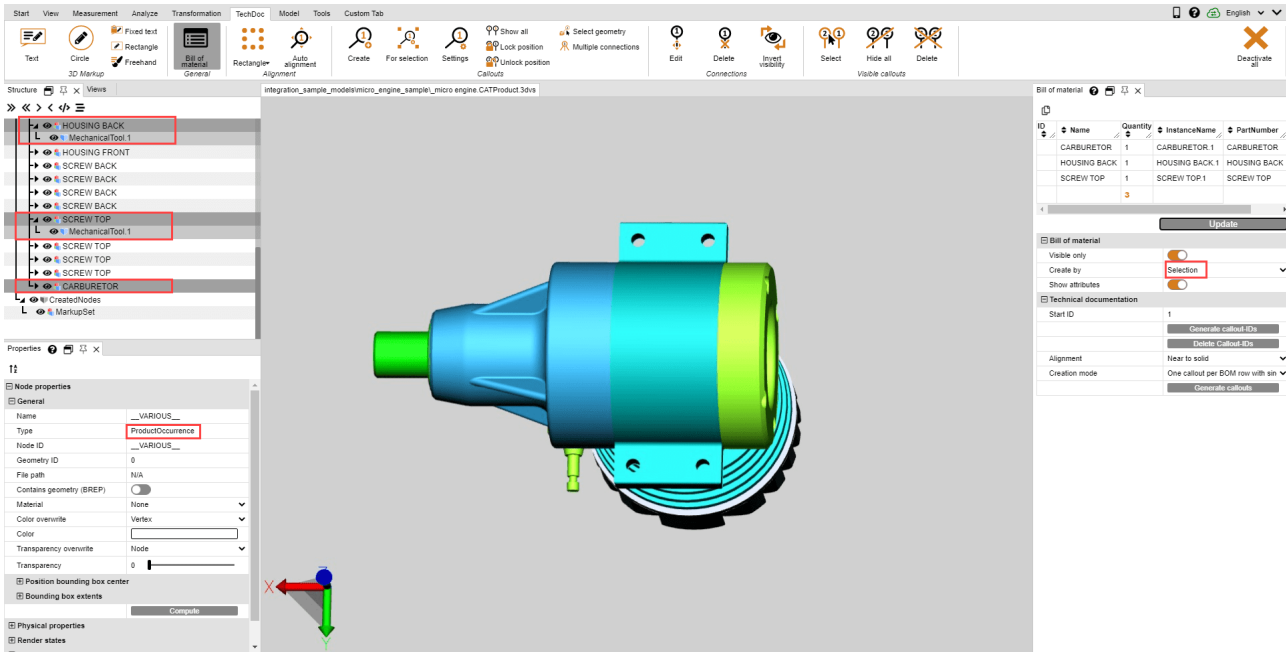
This bill of material contains the number of lowest product occurrences in the structure (type **PRODUCTOCCURRENCE**).



Selection


This bill of material contains the number of the highest structure level of a selection. This allows you to create a bill of material on any hierarchy levels for more complex models.

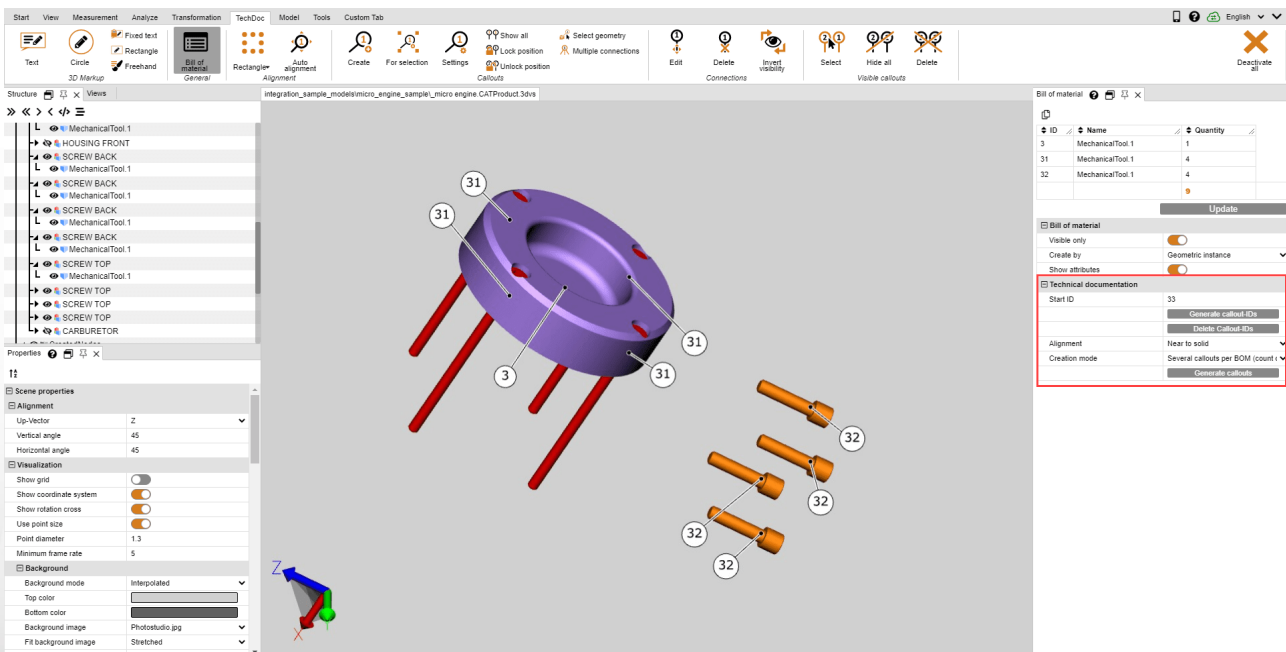
Note : A selection at a specific structure level can easily be created for the entire model. To do this, select the desired structure level and then press the last button under the title bar of the structure window () to select all nodes of the same level.



1.25.2 Callouts

A callout is a textual markup element that is connected to a part of your model by a line, an arrow or a similar graphic.

Based on a previously created BOM , callouts can be generated and positioned automatically. You can do this using the functions in the **TECHNICAL DOCUMENTATION** category of the **BOM** tool pane.



The process for generating callouts always consists of the following actions:

1. Creating a bill of material (see the chapter [BOM](#) ⁴⁵)
2. Generate callout IDs for the individual elements of the BOM (see below)
3. Create the callouts in the model (see below)

Generate callout IDs

To display callouts in the model, IDs must first be generated for the BOM lines to identify the callouts. These can be automatically generated for the entire bill of material by clicking the **[GENERATE CALLOUT IDS]** button.

Clicking the **[DELETE CALLOUT IDS]** button these are removed again. You can also enter, change, and remove individual IDs manually.

Note:

- When you update the BOM, all previously generated IDs are removed.
- The IDs begin with the value stored in the **START ID** field. This is incremented as soon as callout IDs have been created. This means that no duplicate IDs are generated for new bills of material. You can manually set the Start ID to a desired value.

Generating Callouts

If callout IDs have been generated as described above, you can now automatically add callouts to the model. Click on the **[GENERATE CALLOUTS]** button.

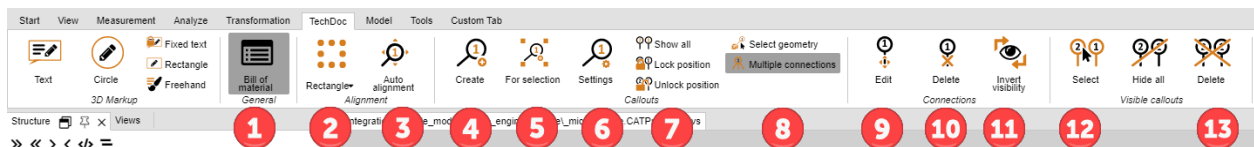
There are two settings available for generating callouts: alignment and creation mode.

The **ALIGNMENT** setting determines whether the callouts should be displayed, for example, in a rectangle around the entire model or only at the top.

The **CREATE MODE** setting allows you to specify whether a callout is to be created for each geometry or, as in the example, a callout with one connection per screw, etc.

Editing Callouts (Description of Menu Options)

Using the **TECHDOC** tab, you can create further callouts, edit existing ones and delete them. These functions are available to you in detail:



"BOM" section

	Element	Description
1	BILL OF MATERIAL	Use this function to open or close the BOM window.

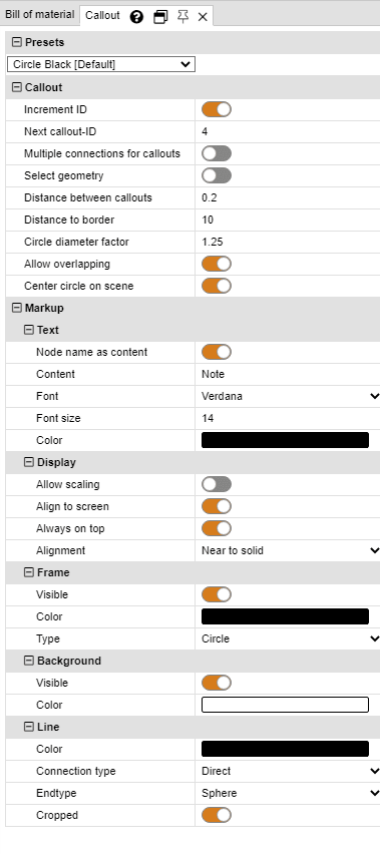
"Alignment" section

	Element	Description
2	RECTANGLE	You can use this function to select how callouts already created should be aligned (e.g. as a circle or rectangle around the model, top and/or bottom, left and/or right of the model, etc.)

	Element	Description
3	AUTO ALIGNMENT	When you realign the camera, created callouts are automatically adjusted to the new perspective. If you disable this option, the callouts remain in their position.

"Callouts" section

	Element	Description
4	CREATE	<p>You can use this tool to create individual callouts to a geometry.</p> <p>Note:</p> <ul style="list-style-type: none"> ▪ If you click on a geometry without a callout, a callout with ID 1 is created or incremented by 1. ▪ If you create a callout on a geometry to which a callout already points, it will be replaced and incremented by 1. ▪ If you create a callout on a geometry with multiple connections (see 8), the connection of that geometry is deleted and a single callout is created. ▪ You can read and set the counter for the next ID in the callout settings (see 6).
5	FOR SELECTION	<p>Use this tool to create callouts for all geometries of a selection.</p> <p>Note:</p> <ul style="list-style-type: none"> ▪ A callout is generated for each selection. ▪ If the MULTIPLE CONNECTIONS function is activated (see 8), a callout with one connection per geometry is generated.
6	SETTINGS	Use this button to open the settings window for callouts. Here you can define settings for the behavior and visualization of callouts.

	Element	Description
		
7	SHOW ALL	This function displays all hidden callouts. The visibility of markups and PMIs is not changed.
	HIDE ALL	This function hides all visible callouts. The visibility of markups and PMIs is not changed.
	LOCK POSITION	This function fixes the callout position for manual and automatic alignment. All visible or selected callouts are locked. This property can be found in the Properties window under the CONSTRAINTS category as soon as a callout has been selected.
8	UNLOCK POSITION	This function allows the manual and automatic alignment of callouts again. All visible or selected callouts are unlocked. This property can be found in the Properties window under the CONSTRAINTS category as soon as a callout has been selected.
	SELECT GEOMETRY	You can activate this check box to select the referenced geometry/geometries when selecting a callout. Accordingly, when selecting a geometry, the callout and all other geometries referenced by this callout are selected.
	MULTIPLE CONNECTIONS	If you select several geometries and create callouts for this selection (see FOR SELECTION function (5)), you can specify whether a callout should be created

	Element	Description
		for each geometry or a callout with connections to all selected geometries.

"Connections" section

	Element	Description
9	EDIT	<p>If you have already created callouts, for example, using the bill of material (BOM), you can use this function to subsequently edit their connections. For this, activate this function first. You then have the following options:</p> <ul style="list-style-type: none"> You can click on a geometry already referenced by a callout to reposition its connection. The callout is also selected hereby! You can select a callout and then click on a geometry to change the position of an existing connection or create an (additional) connection. Note: If necessary, a connection of another callout is removed, because a geometry can only be referenced by one callout!
10	DELETE	<p>You can use this function to delete existing callouts. For this, activate this function first.</p> <ul style="list-style-type: none"> If you click on a callout or its connection, this callout is deleted. If you click on a referenced geometry of a callout, the connection is deleted. The callout is also deleted with the last connection.
11	INVERT VISIBILITY	<p>You can use this function to invert the visibility of the callout. For this, activate this function first.</p> <ul style="list-style-type: none"> If you click on a callout or its connection, all connections are shown or hidden. If you click on a referenced geometry of a callout, only this connection is shown or hidden.

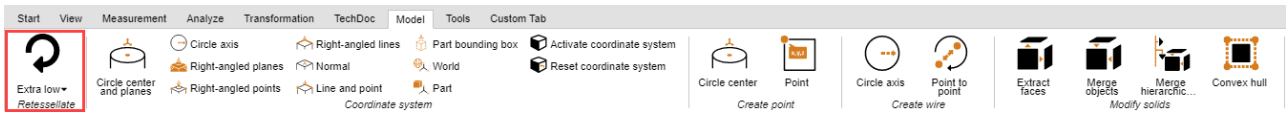
"Visible Callouts" section

	Element	Description
12	SELECT	You can use this function to select all visible callouts.
13	DELETE	You can use this function to delete all visible callouts.

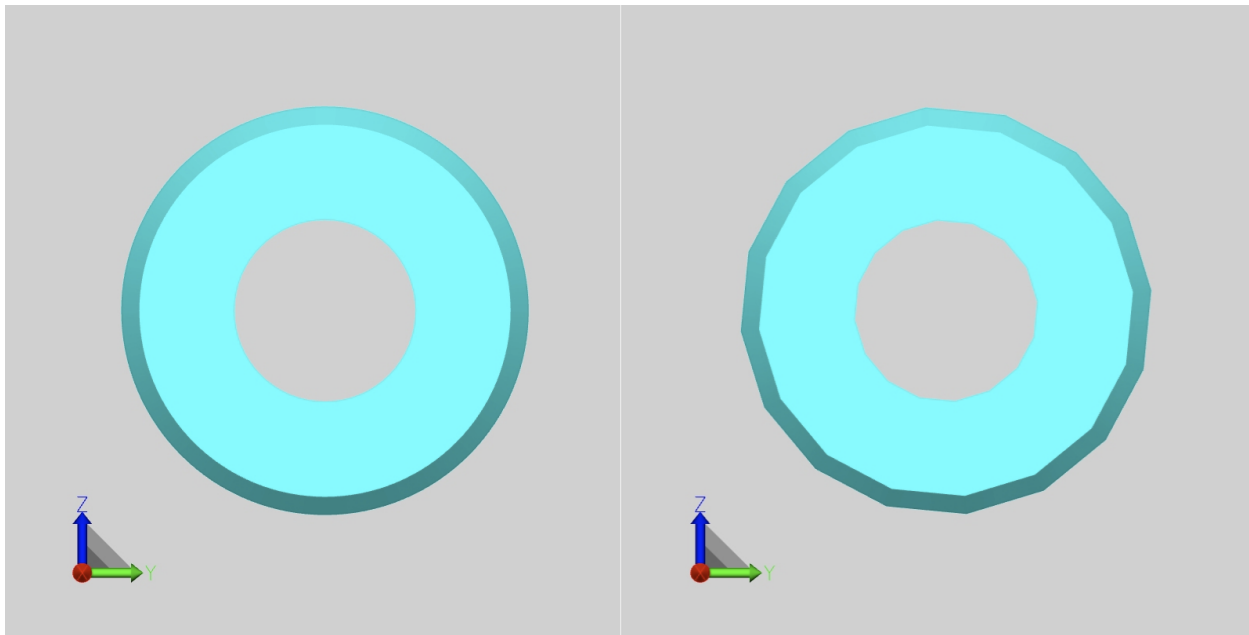
Note : Individual callouts can be edited as usual using the structure or the selection mode in the **START** tab.

1.26 Node Retessellation

The button **RETESSELLATE** offers you the possibility to change the quality of the tessellation afterwards, provided that geometry data (BREP) are part of the model (see chapter [View and Edit Properties of Selected Objects](#)^[8]).



- Select the node(s) in the structure or the relevant geometries in the model area to adjust the tessellation quality.
- As a next step, click the **RETESSELLATE** button in the **MODEL** tab and specify the desired tessellation quality. Alternatively, you can right click your selection in the structure or the selected geometries to retessellate via the context menu item.
- Specify the custom quality setting in **SETTINGS** under **IMPORT SETTINGS**. Additional information can be found in the info box.



Difference between a very high (left) and very low (right) tessellation quality.

Settings

Search Settings...

Default | Current scene

Import

General

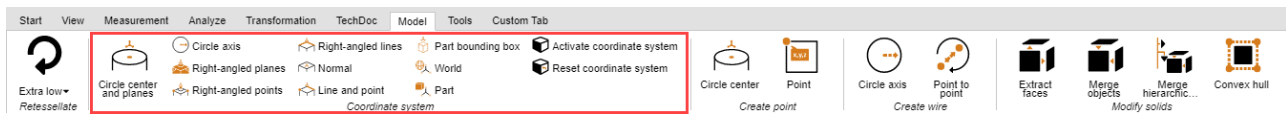
Import mode	Prefer 3D	▼
Prefer tessellation or geome...	Prefer geometry (BREP)	▼
Level of detail of tessellation	User defined	▼

User defined tessellation

Maximal tessellation chor...	0.1
Chord height ratio	50
Angle tolerance	20
Accurate tessellation	<input type="checkbox"/>

1.27 Generate Coordinate System

3DViewStation allows you to generate and activate custom coordinate systems to use as reference for various functions (e.g. for [Transformation](#) ³²).



You can use various methods for the generation of coordinate systems. Each option displays a preview of the coordinate system:

CIRCLE CENTER AND PLANES: Select a circle into the center of which you want to place the coordinate origin. Next, click to select at least 2 planes perpendicular to each other. A preview of the coordinate system is displayed. Click the background to complete the creation. You didn't select enough planes if no preview is shown when you position the cursor on the background.

CIRCLE AXIS: As a first step, select the desired circle and specify the alignment of the remaining axes with another click.

RIGHT-ANGLED PLANES: Select 3 planes perpendicular to each other. Clicking a previously selected plane again will invert the current axis.

RIGHT-ANGLED POINTS: The first point denotes the origin, the second point the x axis and the third point the y and z axis. Lines and edges are captured to generate e.g. a coordinate system on the corners of rectangular solids.

NORMAL: Generates a coordinate system vertically on a plane.

RIGHT-ANGLED LINES: With a method similar to Right-angled points, you can generate a coordinate system by selecting 3 consecutive lines at right angles to each other.

BY LINE AND POINT: In a first step, select a line specifying the z axis. You can then specify the remaining axes with another click (point).

PART BOUNDING BOX: Generates a coordinate system at the center point of the part bounding box of a geometry. Activate the function first, and then select the part.

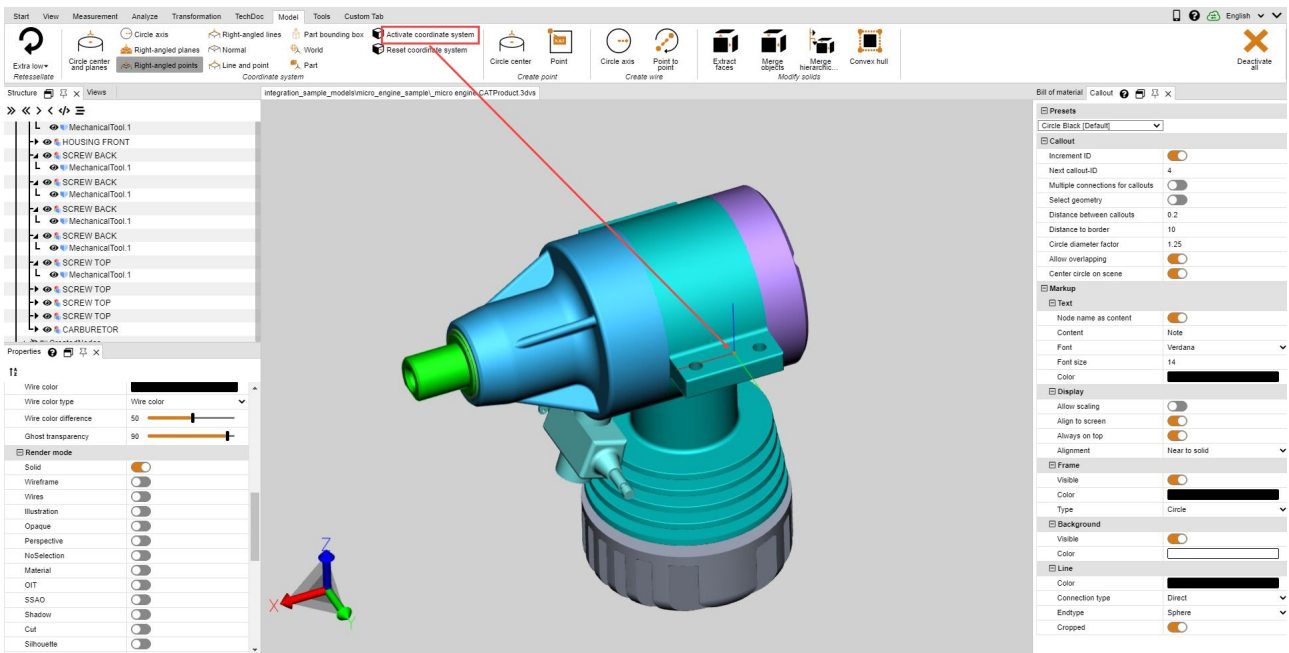
WORLD: Generates a coordinate system at the center point of the part bounding box of a world.

PART: Generates a coordinate system at the origin of the selected part.

ACTIVATION of a coordinate system: A generated coordinate system must be activated to allow its use as reference for e.g. measurements. To activate a coordinate system, it must first be selected in the model area or in the structure, followed by an activation via the button **ACTIVATE**. An active coordinate system is displayed in color. The position of the coordinate system visualization in the bottom left corner of the model area is adjusted in accordance with the active coordinate system.

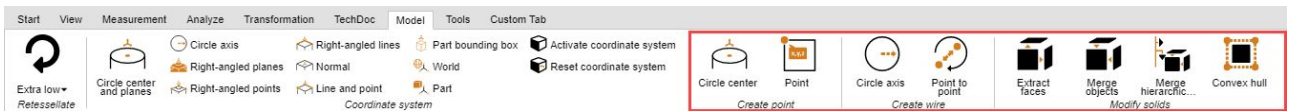
Click **RESET** to revert to the original world coordinate system.

You can automate the activation of a generated coordinate system with the setting **ACTIVATE AFTER CREATION**. The sub option of this setting allows the generation of a cut in the specified alignment axis for this coordinate system.

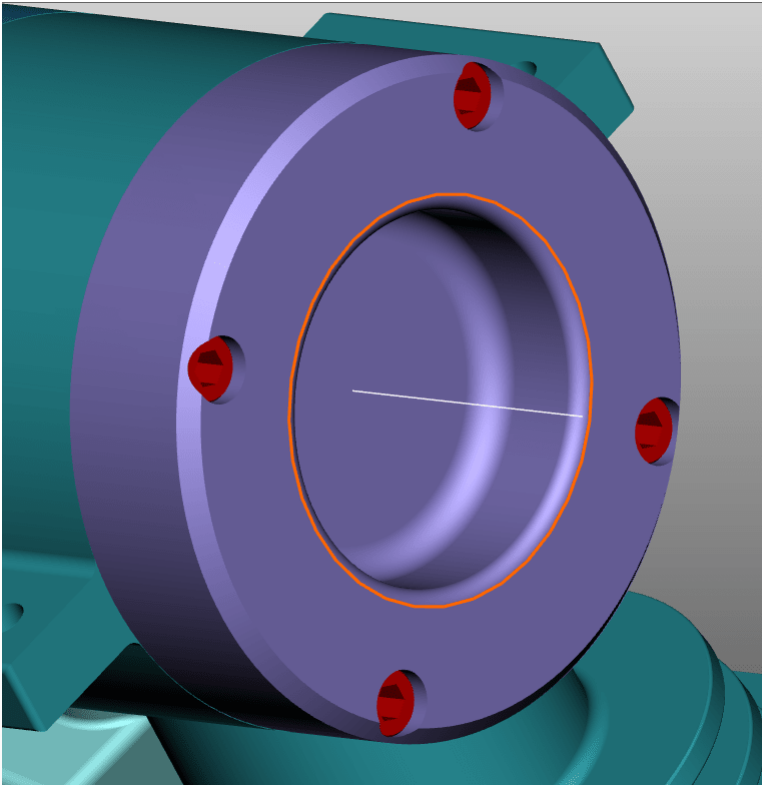
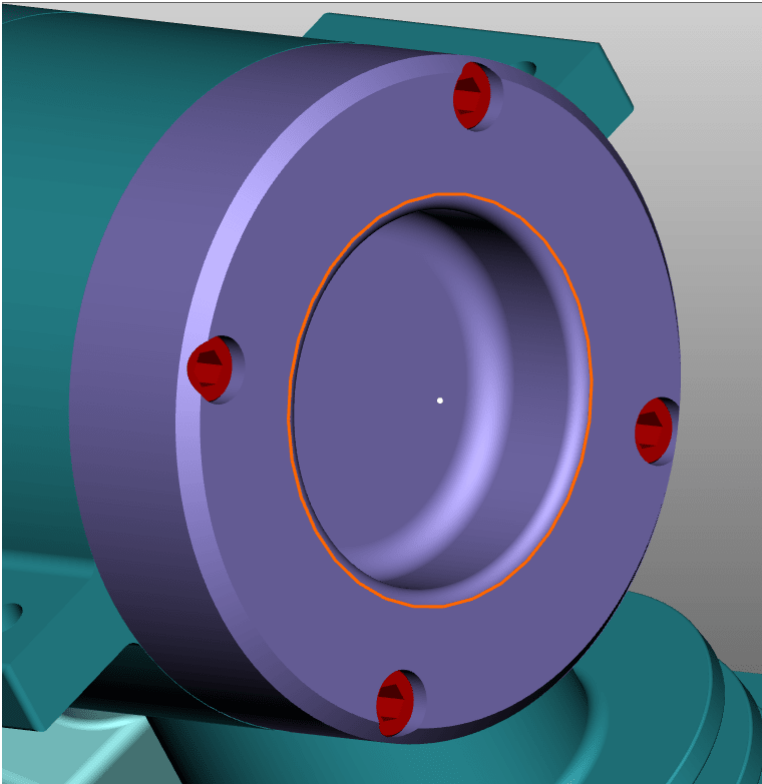


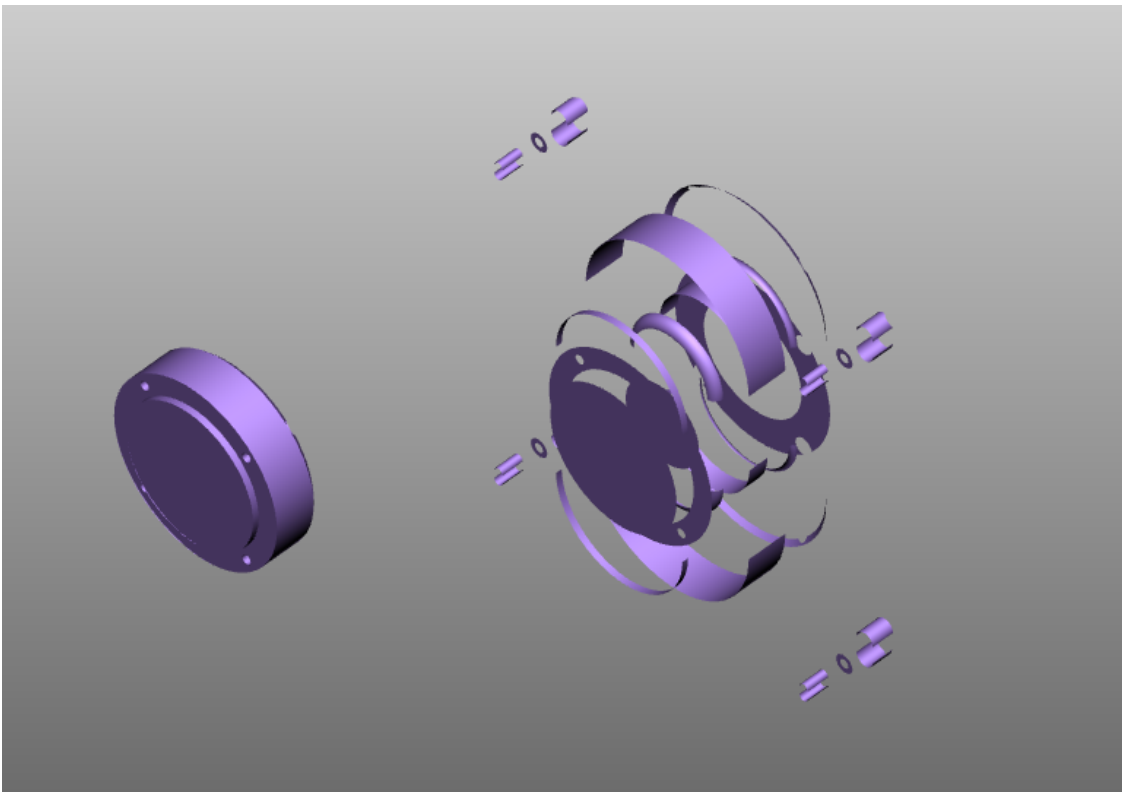
1.28 Generate Auxiliary Geometries

Auxiliary geometries like points, lines or areas are frequently needed for the analysis of parts. If these are not available, you can use the following auxiliary geometries to generate them.

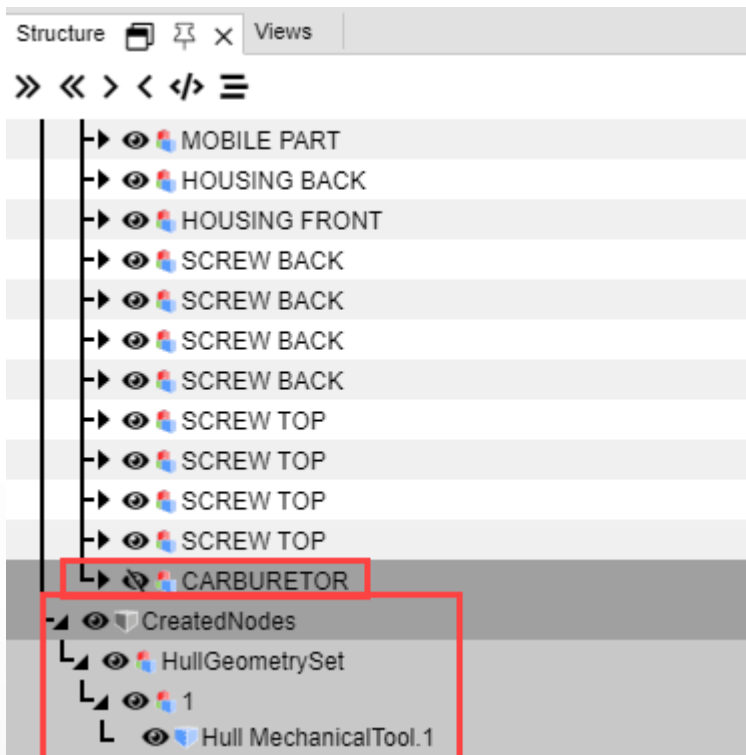


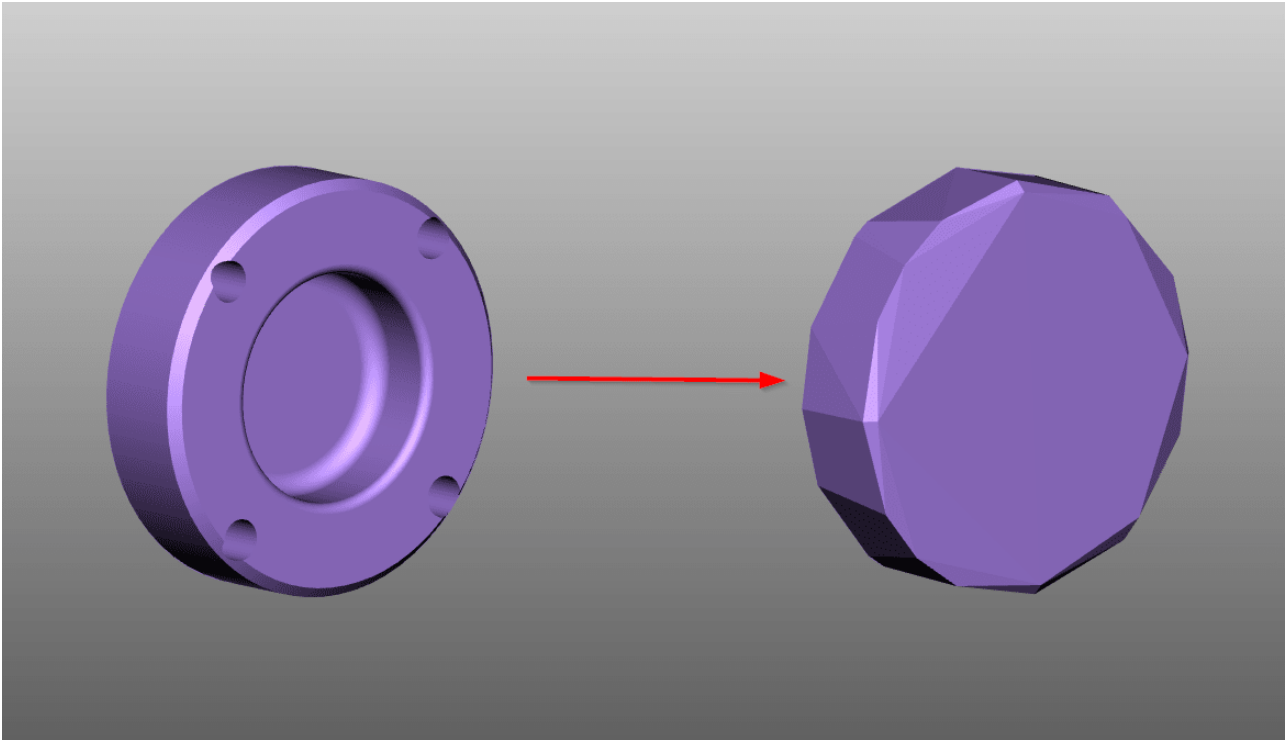
Function	Description
CIRCLE CENTER	Generates the circle center point via the selection of a circle segment (illustration on the left).
POINT	Generates a point at any selected location of a geometry.
CIRCLE AXIS	Generates a circle axis via a circle segment (illustration in the middle).
POINT TO POINT	Any two points of a geometry can be selected to generate a line between two points.
EXTRACT FACES	Generates a surface model from a solid model. Select the geometry and decompose it into surfaces via the function EXTRACT FACES (illustration on the right).





CONVEX HULL: Distorts a geometry by generating a hull. The original geometry is hidden.





All auxiliary geometries created by 3DViewStation are saved in the structure below the node **CREATEDNODES**. One exception is geometries generated via the function **EXTRACT FACES**. These will be placed directly below the relevant product node with the suffix **FACES**.