Preparation of Assembly Drawings

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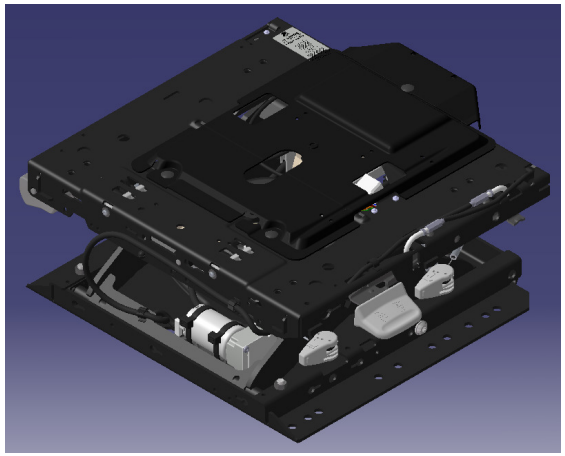
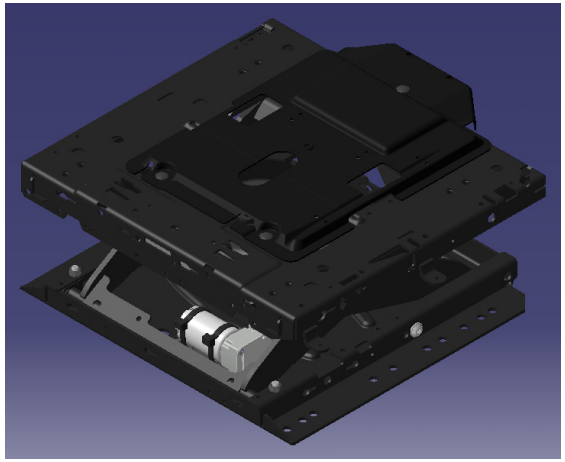
7th May 2012

Rev. 08

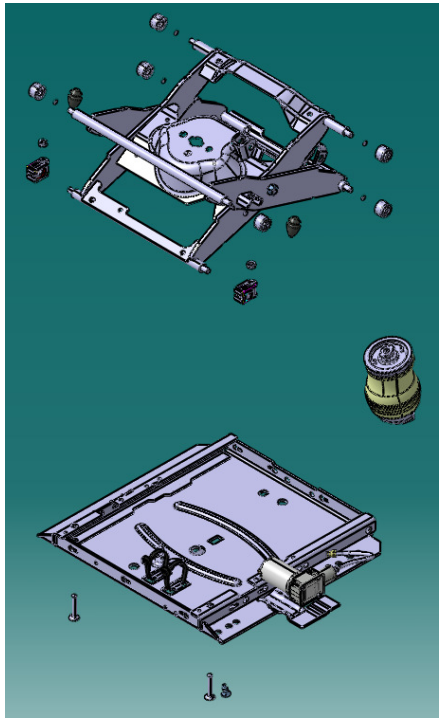
Complete 3D model



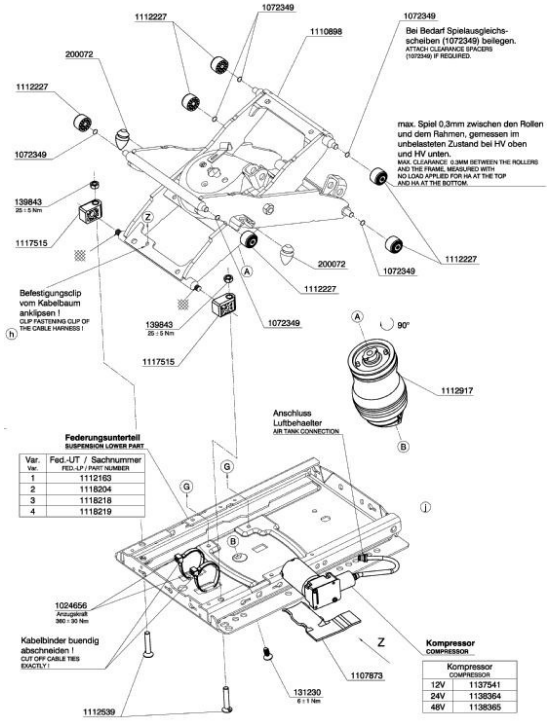
3D_MZ model



Scene – exploded view



Drawing



Specifications

Objectives:

Definition of a uniform procedure regarding the preparation of assembly drawings (German abbreviation "MZ") taking into account the following aspects:

- from the complete 3D model to the assembly drawing
- creating an enhanced scene (exploded view) as a template for the assembly drawing to be created afterwards
- basic settings of the drawing

Important functions	Enhanced Scene
Components	An assembly is required.
Specific characteristics	none
Requirements	Basic knowledge of CATIA V5, preparation of drawings (Drafting) with CATIA V5

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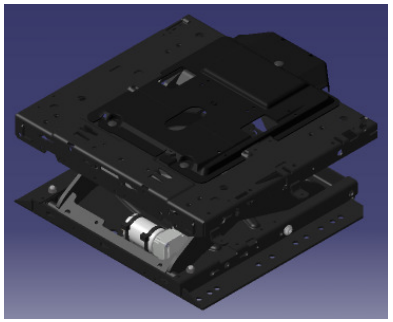
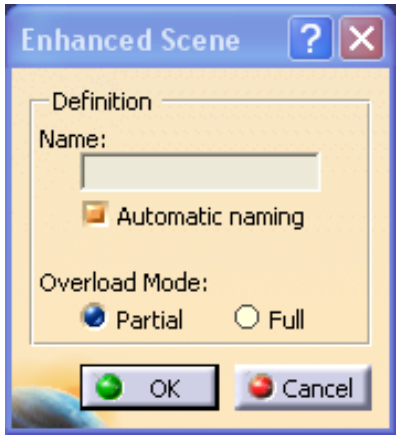
Amberg, 09/2005

Creating an Assembly_MZ, Creating a Scene



Combined _MZ assembly



Create an assembly X XXX XXX_MZ. Insert the assembly to be mounted (e.g. FB_XXXXXXX = functional unit) and its adapter parts into the newly created assembly.

Create a scene by means of the "**Enhanced Scene**" button (= basic setting). In a new window, the scene is named automatically. Deactivate the automatic naming to insert the name manually. For the "Overload Mode", select "Partial".

Click OK to open the scene (green background). Here, a new scene-specific toolbar is displayed.

Scene-specific toolbar

Description of the "Enhanced Scene" Toolbar

"Explode"

Exploding an assembly in the scene. The changes will not influence the model.

"Save Viewpoint"

Saving the view in the scene.
Enable this function to display always the same view when activating the scene.

"Apply Scene on Assembly"

The changes made in the scene will also be applied to the assembly.

"Apply Assembly on Scene"

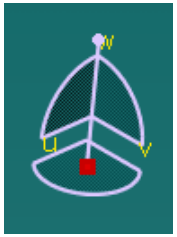
The changes made in the assembly will be applied to the scene. This can also be used to undo changes of the scene and to align them with the model.

"Exit Scene"

Select the scene in the structure tree to activate it again.

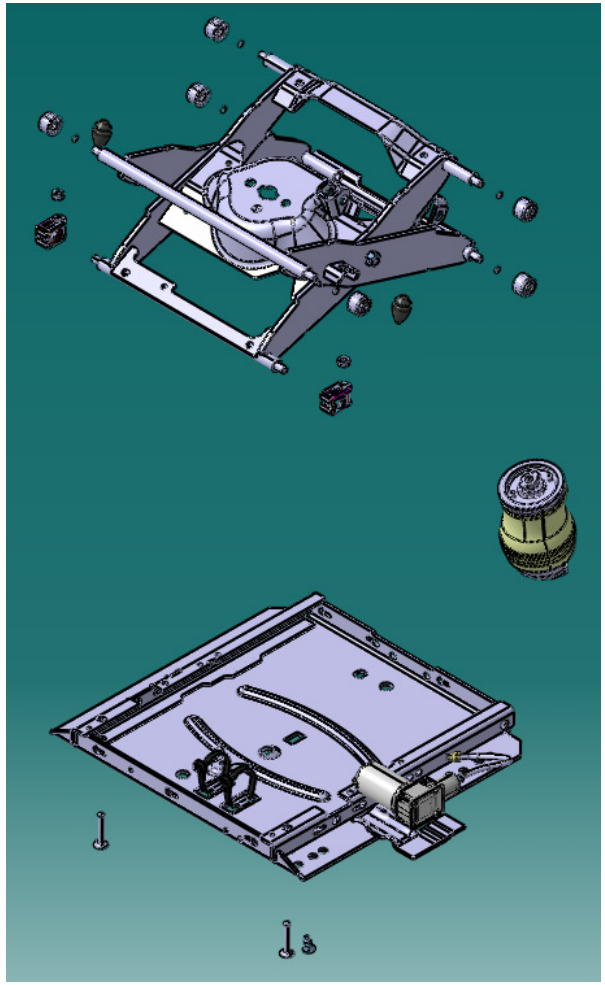
Creating an Exploded View (Moving Components)

3D compass

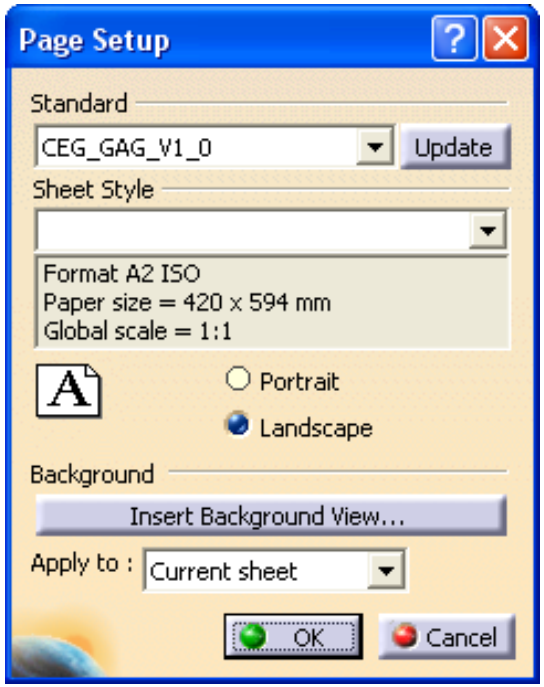


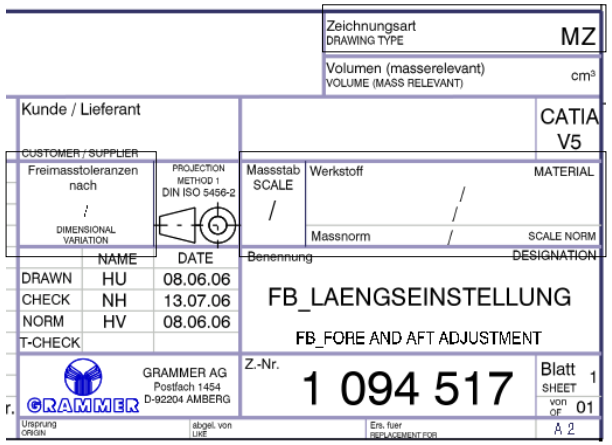
- Moving of components (rough positioning)
- Select the compass at the red point,
  drag it to the part and release it there.
- The compass is active when it is shown in green colour. Only then it is possible to move parts.
- It is possible to move several parts at the same time (in the tree diagram or in 3D), if they are selected with the Ctrl key being pressed.

Components moved by means of the compass



Basic Settings of the Drawing





Choose the basic settings of the drawing under "**File -> Page Setup**".

As "Standard" setting, select "AZ_MZ_GAG_V1_0" and for "Sheet Style“ select the respective sheet size to make sure that all line widths are available for selection.

Filling in the profile card for the title block of the drawing:

Drawing type: MZ (German abbreviation for "assembly drawing")

Material: /

Scale: /

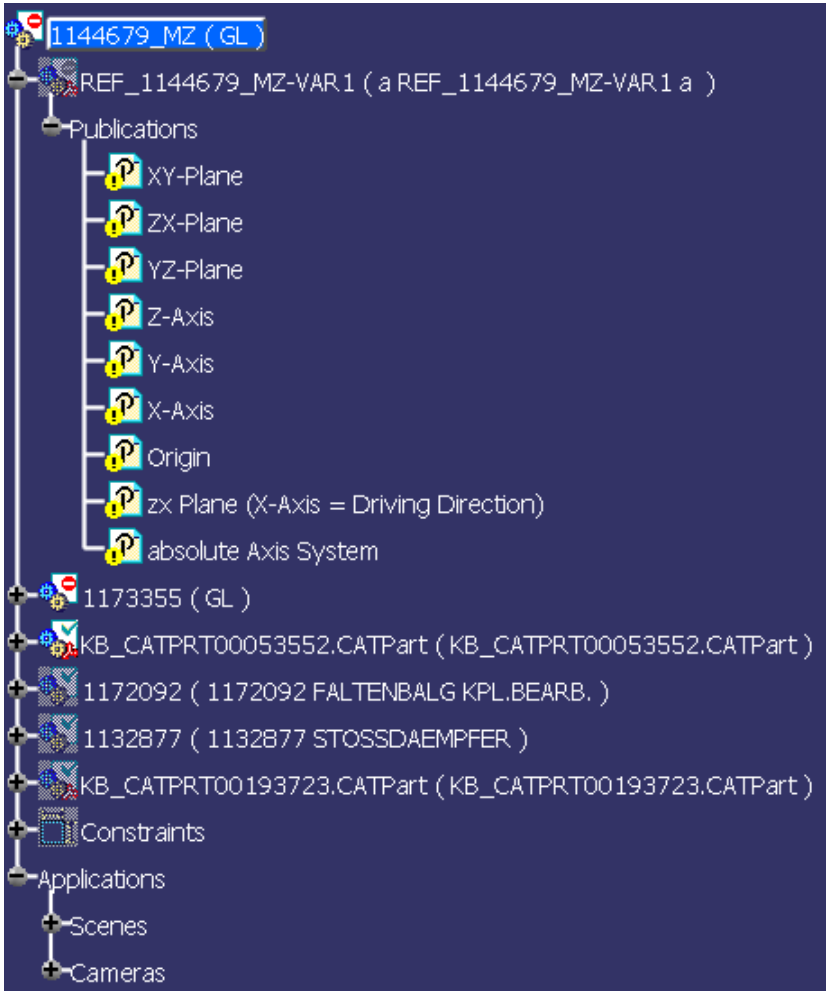
Dimensional variation: /

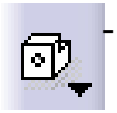
SAP material type: "ZCHN" (drawing)

"**Run Script01**"

The title block is filled in automatically.

Generating a Drawing View

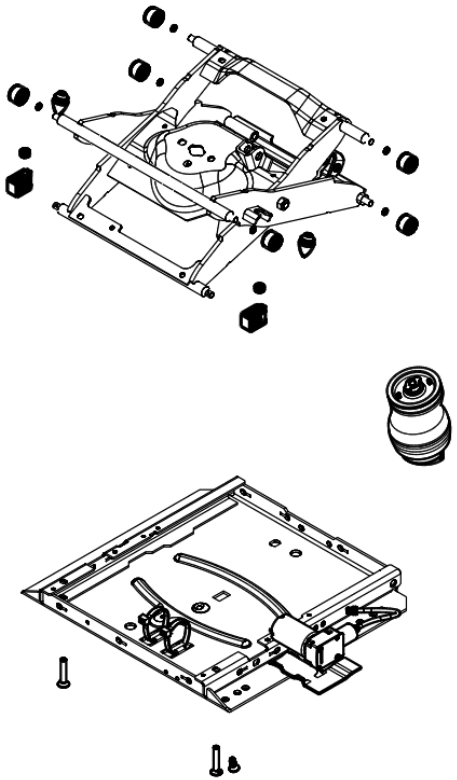




- In the drawing, select the "**Isometric View**" icon and then go to the assembly.

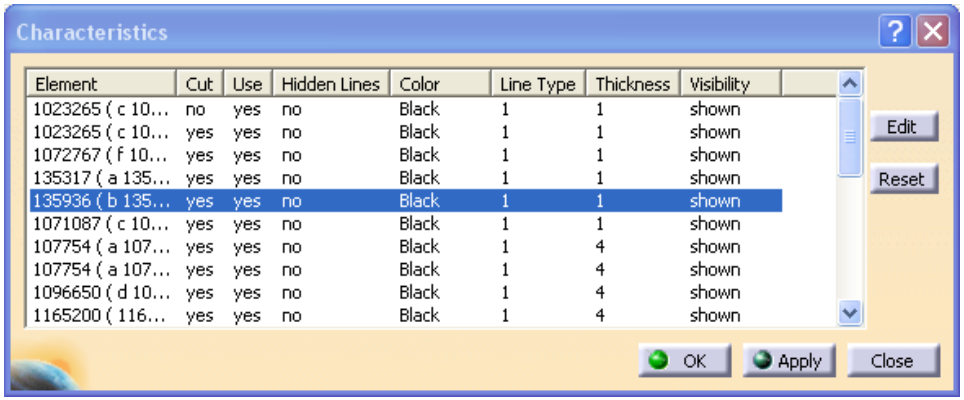
-Click on the scene and select under "REF_" of the respective assembly the publication "Origin".

- The scene is projected into the drawing.

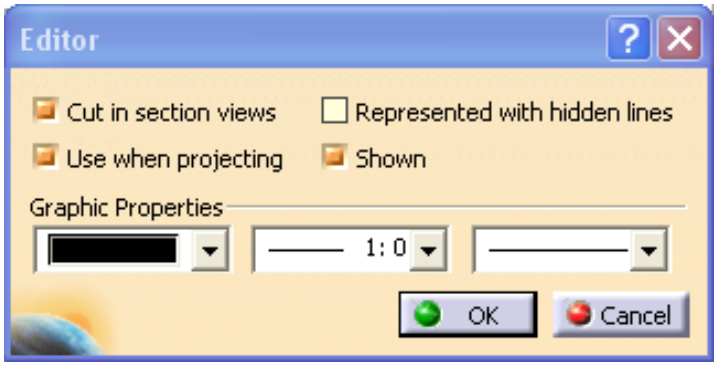


Changing the Line Widths

As besides the components also adapter parts are displayed for assembly drawings, different line widths are used.

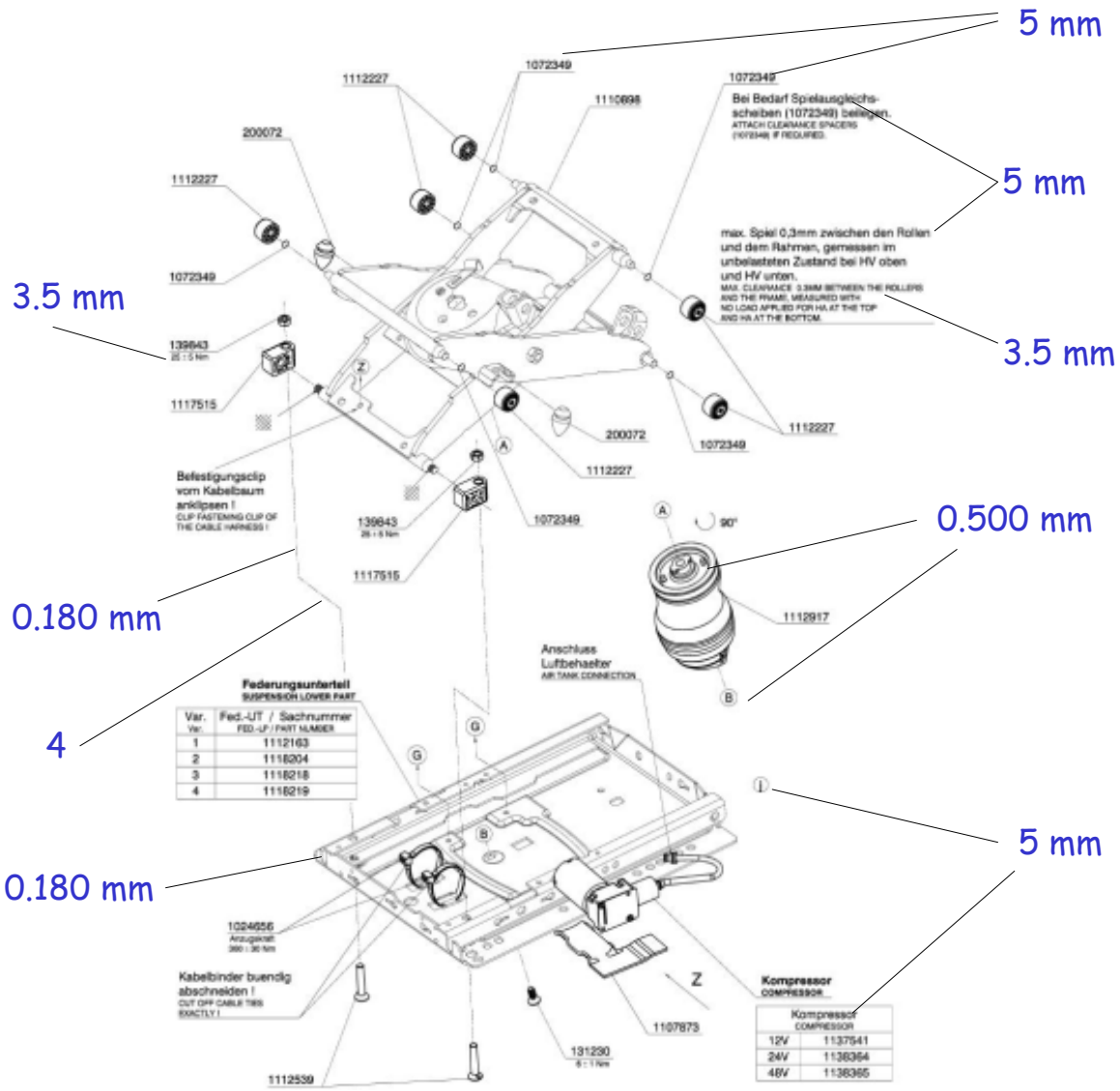


Activate the view frame, click the right mouse button at the frame, select **Object Isometric View -> Attributes Overlay**. The "Properties" window opens.



Select the parts to be modified in the drawing. Selected parts are displayed in the table where they have to be selected once again. Click the "**Edit**" button to open the "Editor" window in which you can change the properties of the parts.

Line Width / Font Size



Line width

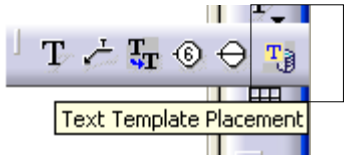
Components:	0.500 mm
Adapter parts:	0.180 mm
Axes:	0.180 mm
Line type:	4

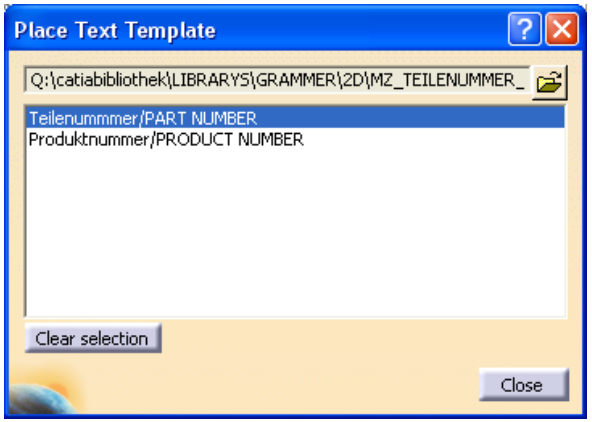
Font size (equal for all frame sizes)

Inventory numbers::	5.0 mm
Reference numbers:	7.0 mm
Tightening torques:	3.5 mm
German texts:	5.0 mm
English texts:	3.5 mm

(Texts in drawings are displayed in German and English and their size remains the same irrespective of the frame used!)

Automatically Generating Part Numbers and Axes





The inventory numbers of the individual parts are added automatically by means of the "**Text Template Placement**" command.

An inventory number must be assigned to **each** part (at least with a line). Quantities such as e.g. 111320 (4x) do not have to be added anymore. Tightening torques are written directly below the inventory number. For this reason, the tightening torque legend is omitted.

When using it for the first time, the text template catalog is selected:

Q:\catiabibliothek\LIBRARYS\GRAMMER\2D\MZ_TEILENUMMER_ERZEUGEN.catalog

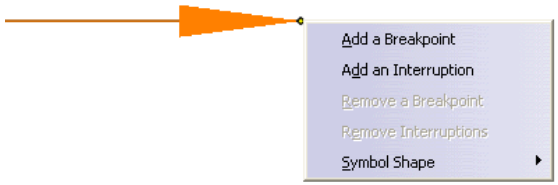
After clicking on a component, the selection of the part number and product number is displayed in the window. !!! Always select the part number only!!!

Make sure to select an edge of the part before, because it is not possible to select an area.

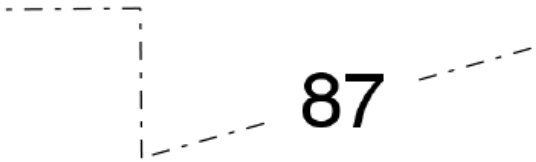
The command cannot be used for assemblies, as the assembly number must be entered manually.

The inventory number is located above the reference line and ends without a symbol shape.

Before: only a simple arrow



Afterwards: axis with interruption and breakpoint



The axes are generated by means of the "Arrow" command. To make sure that the arrow becomes an axis, right-click a yellow quadrangle at the line,

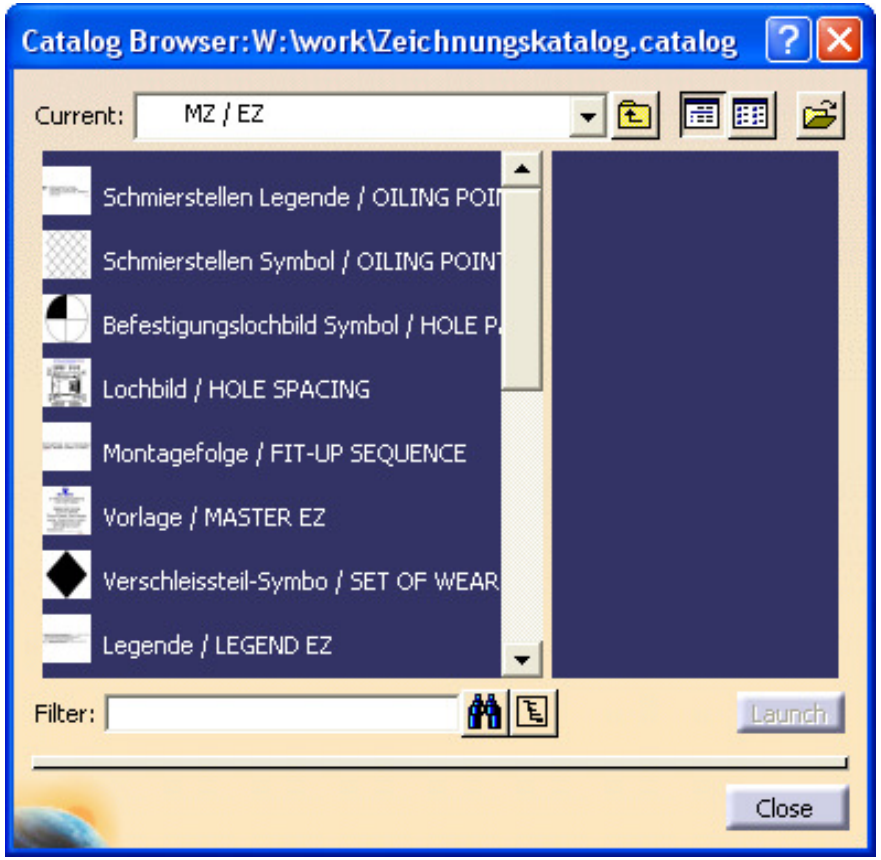
set ***No symbol*** for the "Symbol Shape" and set ***Line type 4*** in the lower bar.

The arrow is used to be able to add interruptions and breakpoints and to

be able to easily modify the axis.

Drawing Catalog

It is possible to insert legends, symbols and texts from the "MZ" folder.



Catalog browser: drawing catalog

(German: "Zeichnungskatalog") for "MZ"

Oiling points legend

= Schmierstellen/OILING POINTS
Sachnummer/PART NUMBER: 1 099 937
Schmierfett/LUBRICATION GREASE KFHC2N-50
DIN 51825

Oiling points symbol

Montagefolge laut Arbeitsplan
ASSEMBLY SEQUENCE ACCORDING TO SCHEDULE
Version 1.1