

# ccAssembler 3.1 – Getting Started

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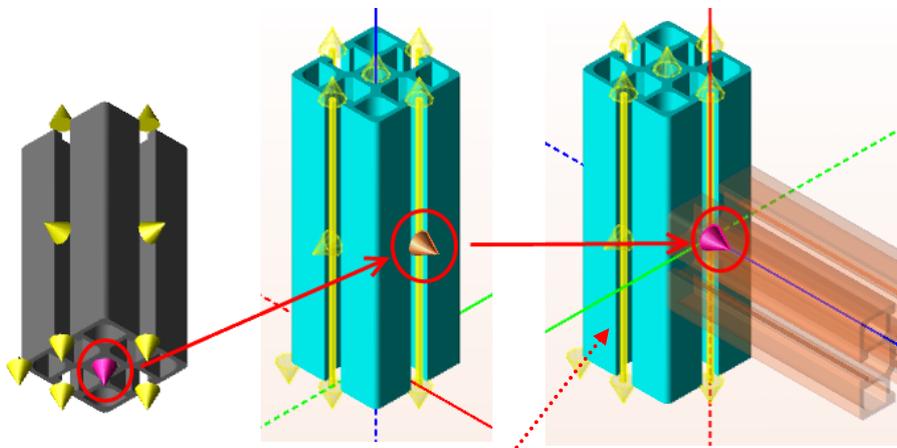
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## 1 BASIC PRINCIPLES

Use of the **ccAssembler** is kept as easy and intuitive as possible. An understanding of a few basic principles is sufficient to experiment and obtain your first results.

### 1.1 INSTALLING ANCHOR ON ANCHOR

A special property of the program is that the actual geometry of the parts is of secondary significance for most of the actions. The **use is instead concentrated more on the snap points defined in the parts**, which are also called **anchors**. They play a decisive role especially when positioning parts in the assembly. In the majority of cases a part to be installed (referred to in the following as "insert" for short) is attached to an already positioned part (referred to in the following as "attachment" for short). To this end, **an anchor is defined in both the insert and in the attachment and the new part is inserted so that the two anchors are congruent**. Rotation about this point is defined afterwards, but often the suggested orientation is the most sensible. This is because the anchors are **preferably installed in opposite directions to each other, i.e. "tip to tip"** (provided the parts logic does not insert additional rotations, but more on that later).



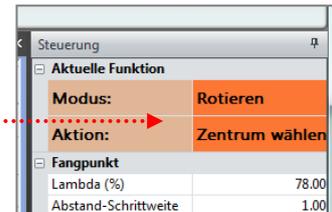
The so-called "**line anchors**" play a special role; these not only represent a specific position, but also an **arbitrary position on a line** (e.g. along a groove). The precise position can be determined either by orientating it **on the distance from the end points** or as a **percentage lambda value** (e.g. 50% for the middle point between the end points).

The anchors are not only used for the insertion of parts but also as **reference points**, e.g. for measuring or as the **centre for rotations**.

## 1.2 MODES AND ACTIONS

The main user actions take place via the program's Main window. **By choosing a command (in the menu or context menu), this main window is placed in a corresponding mode, which defines a sequence of actions.**

The current action in turn determines the consequences that a mouse movement, mouse click or pressing a key have. **The currently active mode and the current action are always displayed at the top in the Control window.**



For example, if you want to rotate and already positioned part, select the "Rotate" command from the menu in order to activate Rotation mode. If a part has not yet been selected, in this mode the action "Select Part" is set first. The part to be rotated can then be selected with a click, after which the next action to be set is the selection of the centre of rotation. Actions for selecting the axis and for defining the angle follow. Following completion of the last action the mode is ended and the program returns to selection mode. There are also modes that do not end automatically, but remain active until you select another mode. This is described in detail further below (Chapter 4, 5).

It should be noted that **cancelling an action** often does not cancel the whole command. Instead, the program **jumps back to a preceding action**, e.g. to select another centre of rotation without having to reselect the parts concerned.

## 1.3 MOUSE CONTROL AND DIRECT INPUT

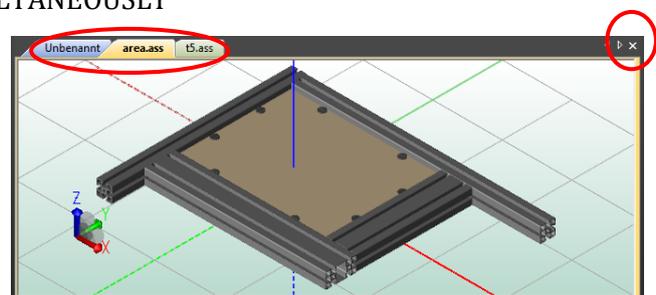
One of the biggest advantages of the ccAssembler is the ability to handle a large part of the user actions using the mouse without having to enter numerical values with the keyboard.

Nevertheless, in several situations it is easier to **enter a value directly**, e.g. if you want to select a specific position on a line anchor. **This can be done via the Control window.** In order to avoid conflicts with the mouse movement, with most actions it is possible to **switch the mouse control on and off using the Backspace key**. In the Control window, the **increment for the mouse control** can often also be defined.

The idea of making functions **available not only via the mouse but also via the keyboard** is continued in that on the **right-hand side of the Main window**, depending on the current action, **buttons can be displayed**, which partly serve as alternatives to keyboard commands. For example, one of these buttons enables the user to switch between local and global coordinates, which can also be performed using the space bar.

## 1.4 EDITING SEVERAL DOCUMENTS SIMULTANEOUSLY

In ccAssembler it is possible for several documents to be open at the same time. This is shown by the different coloured tabs at the top edge of the Main window. Here it is also possible to switch between the



documents. The current document is closed by clicking the "X" in the right of the tab.

## 1.5 UNDO AND REDO

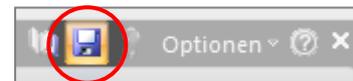
When using the program, the history of most actions is recorded. This means that in case of incorrect inputs it is possible to undo the recent operations step-by-step. As long as no other changes are made, the operations can also then be restored (redo). This function, also called Undo/Redo, can be opened using the buttons in the top left-hand corner of the Assembler window. Alternatively, the key combinations Ctrl+Z and Ctrl+Y can also be used.



## 1.6 AUTOMATIC BACKUP

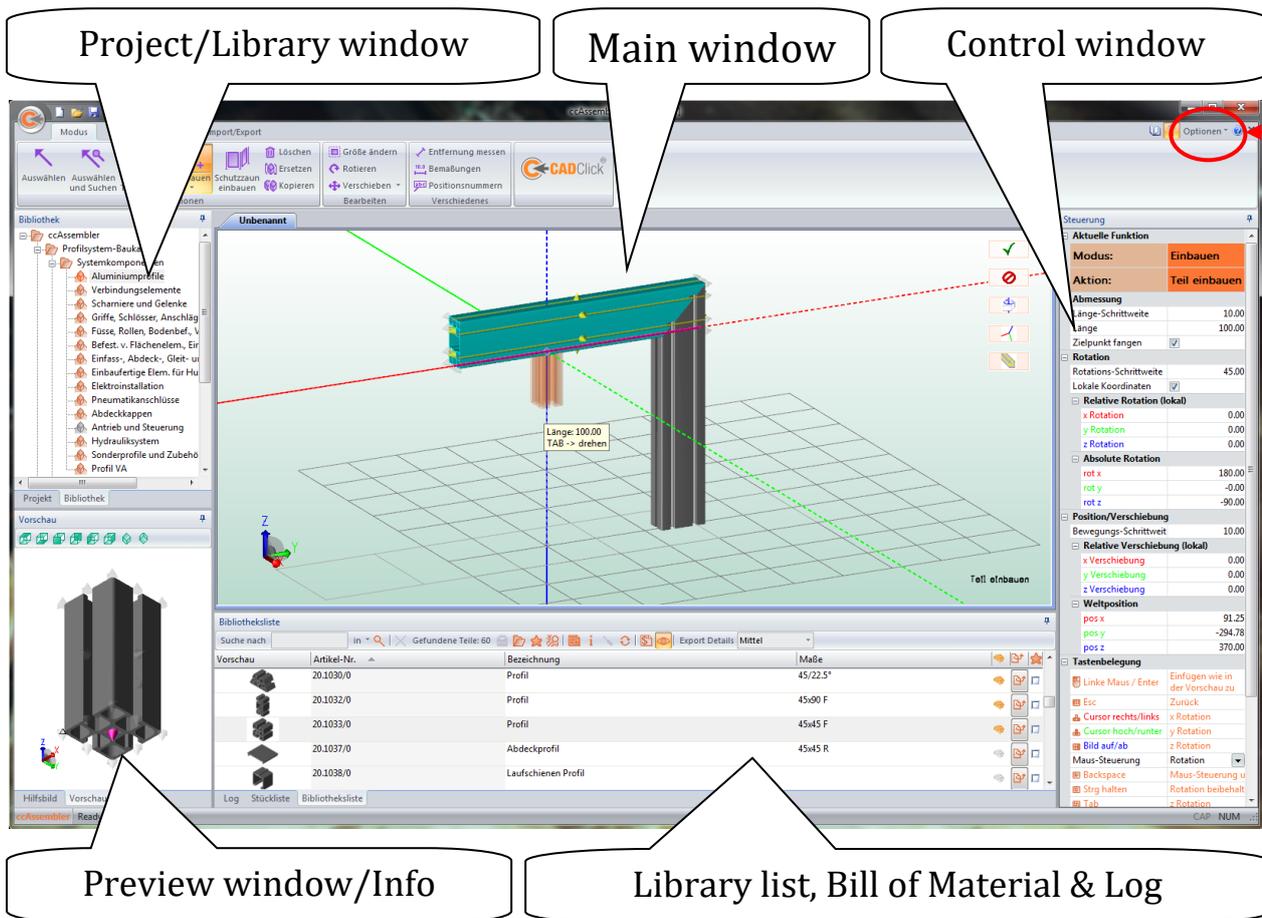
When working with a previously saved Assembler file (i.e. if the file is opened in the program) a backup of this file is created automatically in the background every 10 minutes. If the Assembler is ended unexpectedly (and thus without the opportunity of saving), on opening the file subsequently the user has the choice of opening either the original file or the most recent backup. On closing the file or normal ending of the Assembler, the backups are deleted automatically.

If the automatic creation of backups hinders the workflow, this function can also be disabled. This is done using a switch in the top right-hand corner in the Assembler window.

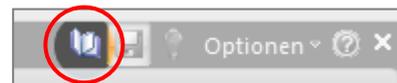


## 2 THE WINDOWS

The user interface of the ccAssembler enables the **individual windows to be ordered any way the user wants and to change their size**. The **starting position** can be restored at any time using the **"Options → Reset Window"** menu command.



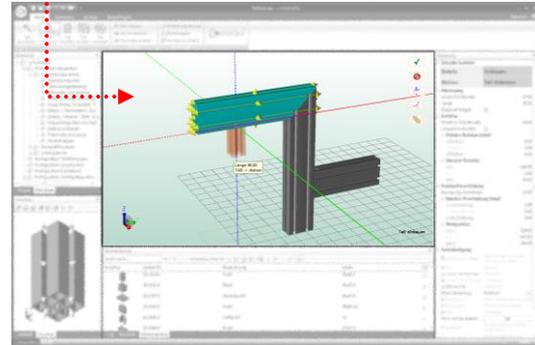
Apart from the Options menu, it is also possible to switch to **Catalogue Mode** (or back). In this case the Main and Control windows and the menu are hidden and the remaining windows are rearranged. This display is particularly suitable if you only want to see parts and to compile a bill of materials without specifically installing them in 3D.



The individual windows are briefly explained in the following.

## 2.1 MAIN WINDOW

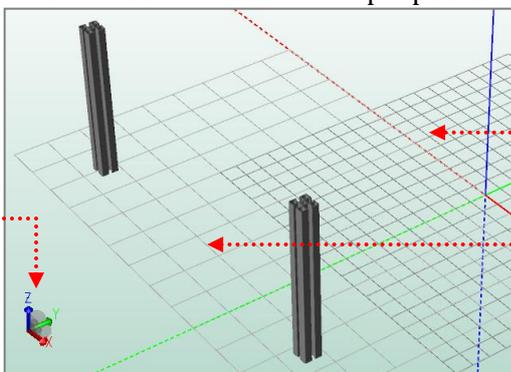
The main window **represents the current assembly and forms the centre for most of the interaction**. The significance of mouse clicks and keys varies depending on the mode selected and the current action. In general, however, **the left-hand mouse button is used for interaction**, the **middle mouse button is used to change the view** and the **right-hand mouse button is used to open the Context menu** (or the Gesture Control, see Chapter 12.1)



The Main window usually displays the global coordinate system as lines, which originate from the zero point. Continuous lines stand for positive axes and broken lines for negative axes. However, this display of the coordinate system is adjusted by many actions. For example, it often jumps from the global zero point to the reference point of an action and is possibly rotated so that it corresponds to the local coordinates of a selected part.

An **additional coordinate system in the bottom left-hand corner of the window** on the other hand always remains oriented according to the global coordinates. This display also enables **adjustment of the view using the left mouse button**. If this button is clicked and the mouse is dragged at the same time, the **view is rotated** or if the **Shift key is pressed and kept pressed it is moved**. Individual clicks on the axes select the fixed views from above, the front, right, etc. (**please note**: Near this coordinate system, conflicts with the current action of a command can occur due to the use of the left-hand mouse button)

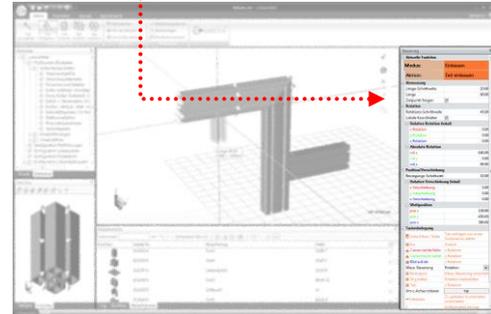
The "View" menu can also be used to show a **grid in the x-y plane**  in the Main window, which is used for orientation purposes. Its dimensions can be defined using the Options (see Chapter 0) In addition, **this grid can also be used as the floor slab**. If the relevant function  is activated, an anchor is positioned in each of the places where lines cross, at which parts can be attached (or which can be used as reference points). **An activated floor slab can also be selected, moved and rotated like a normal part.**



In addition, a **display of the floor**  can be activated, which always extends, regardless of the grid size, so that the current assembly (including zero point) is covered.

## 2.2 CONTROL WINDOW

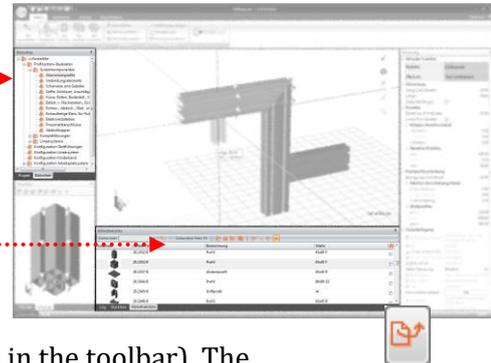
This window fulfils the important function of **always displaying the active mode and the current action**. In addition, the **current interaction options via the mouse and keyboard are listed**. Here it is therefore possible to tell at a glance what is being done, what has to be done or can be done. The window also provides several interaction options in that, primarily, **numerical values can be entered and changed directly** or actions such as the snapping of target points (anchors) can be switched on and off.



## 2.3 LIBRARY AND LIBRARY LIST

These two windows are used to **display the product catalogue and for selection of the parts to be installed**.

The **library window** represents the different product groups in the form of a tree. A group can be selected here, so that it is opened in the library list.



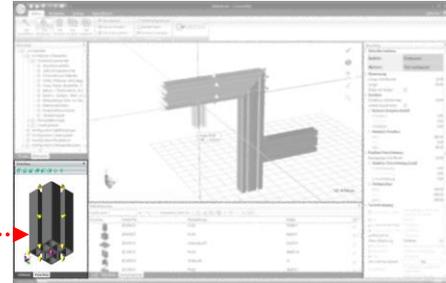
The **library list** lists the products contained in a product group (and possibly their subgroups) together with some information and a preview image. A specific part, e.g. for inserting, is then also selected here. The corresponding button in the entry can also be used to export a part individually (with the degree of detail given in the toolbar). The procedure and options are identical to the export of the whole assembly described later. A list entry can also be dragged into the Main window by means of Drag&Drop to insert the relevant part directly. Alternatively, it is possible to include the selected part in the bill of materials without positioning it in the space. The corresponding button  of the Toolbar can be used for this. It is also possible to insert a part as a sub-part of an already installed part, by dragging the list entry by means of Drag&Drop to the relevant entry in the Project window.

In the Toolbar it is also possible to open an **Information window** with information on the currently selected part. Here there is also the "**Configurations**" function which, if applicable, opens the Configuration window for a selected part (see Chapter 10.5). The "**Refresh Selection**" button is used for a substitute to re-selecting a part in the Library list of the Library tree that has already been selected by clicking it again. In addition, **Previews can be shown and hidden** in the Toolbar.

The list search function is described in Chapter 8.1.

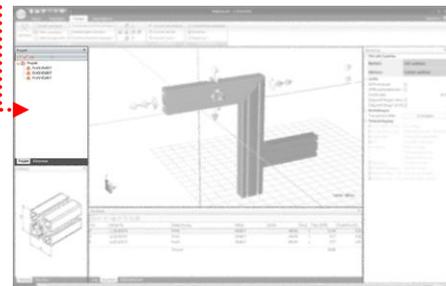
## 2.4 PREVIEW WINDOW

The Preview window displays the part selected in the Library list and if applicable, enables **selection of the anchor to be used for the insertion**. If you keep the **Shift key pressed here while moving the mouse, the part is displayed transparently**, which also makes it easier to select anchors concealed by the geometry. The view can also be changed here using the mouse or Toolbar, which is also the case in the Main window. It is possible to use Drag&Drop to drag the displayed part from this window into the Main window in order to insert the part there.



## 2.5 PROJECT WINDOW

The Project window contains a **tree structure** in which there is an entry for each instance of a part contained in the current assembly. If such an entry is selected the corresponding part in the Main window is also selected and vice versa. For easier orientation it is also possible to **give** each of the individual installed instances **their own name** by **editing the entry**. This is then also used in the bill of materials and is displayed in the Control window, if the part is selected in "Select mode".

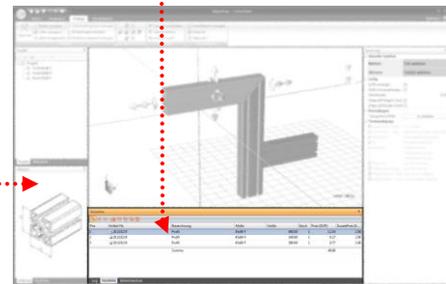


One or several comments  can also be attached to a selected part. These also appear in the bill of material as additional rows under the relevant entry. Selected parts or whole groups (including all parts they contain) can be deleted here . Parts which have been inserted via the Library list and which are not represented in the Main window are coloured grey and can be edited  if necessary. If a part has sub parts, it is marked with a + symbol . The sub parts of the selected part can be edited using the  button.

In addition, instances can be grouped here, which is described in greater detail in Chapter 8.2.

## 2.6 BILL OF MATERIAL (BOM)

A continuously up-to-date bill of material is displayed here and can also be output. This is described in greater detail further below in Chapter 8.3.



## 2.7 INFO

Info on the part currently selected in the library is displayed here (if available).

## 3 CAMERA AND VIEW

### 3.1 MOUSE CONTROL

The **middle mouse button** is reserved for changing the view in the Main and Preview window. By default, holding the button causes a rotation in the form of **orbiting of the camera** around the scene. If the **Ctrl key** is pressed at the same time, a **relative orbit about the clicked point** is selected. (Note: It is advisable to experiment with these two

Middle mouse button	Rotate view
Middle button + Ctrl	Relative rotation
Middle button + Shift	Move view
Mouse wheel	Zoom

rotation options so that you develop an intuition for them, as it is difficult to describe the behaviour). In addition, the scene (or camera) can also be **moved**, by keeping the **Shift key** pressed. The **mouse wheel** is used to **zoom**. Here you should note that the position of the mouse pointer is important (not only when zooming in but also when zooming out).

As already mentioned in the description of the Main window, it is also possible to use the left mouse button to change the view if you are using the **coordinate system in the bottom left-hand corner**. In addition, you can use the "Display" menu to select the **Zoom, Move and Rotate View modes**. These replace the previously active command and enable the respective change with the **left mouse button** to be made anywhere in the window (useful, e.g. if the mouse does not have a middle button or wheel). "**Zoom Window** 

" mode can also be selected as a sub-point of the Zoom function; in this case, the mouse is used to drag a window, which defines the required detail section of the image. A further sub-point here is "Zoom all" in which the zoom level is adjusted without changing the rotation so that the entire assembly can be seen.



The menu also provides the option of quickly switching to default views (from above, underneath, left, right, front and rear or isometric view). Here, too, the detail window is chosen so that the complete assembly can be seen.



## 4 BASIC MODES

### 4.1 SELECTION

#### 4.1.1 *SELECT*

Use this mode to select already installed instances. A **multiple selection** can be made by dragging a **frame**. In general, only parts located **completely in the frame** are selected. To select the parts which are **only partly** within the frame too, you must drag the frame **from the bottom right hand side to the top left hand side** (can be recognised by the violet colour of the frame). In addition, after selecting the first part, **further parts can be added by keeping the Ctrl key pressed**. Equally, parts can also be deselected. If the Ctrl key is pressed it is also possible to add several parts to the selection by dragging a frame.

In this mode it is also possible to define that the selected parts are to be **displayed permanently transparent** (by pressing the T key or the button in the Main window). These parts can then **no longer be selected** in other modes. Therefore, instead it is possible to **select concealed geometry** "through the parts", which can be very useful, e.g. for plates or slabs. However, in this selection mode the relevant parts continue to be selectable, so that the transparency can also be **switched off** again. It is also possible to use the **Control window** to **reset** the transparency **for all parts**.

In this mode (depending on the setting in the Control window), **handles or interactive anchor can be displayed on the selected part**, with which parts can be quickly edited or inserted without changing the mode. This is described in more detail in Chapter 7.

#### 4.1.2 *SELECT AND SEARCH*

In this mode, individual parts are selected (no multiple selection). Unlike the normal Selection mode, the selected part is additionally **selected in the library** (and therefore also in the preview window) **as an insert**, so that further copies of this part can be installed.

#### 4.1.3 *SEARCH FOR FITTING PARTS*

Here not only a part but also one of the anchors is selected. A **search for all parts that can be attached at this anchor is then started** in the Library list (see Chapter 10.6 on Parts Logic). The anchor selection also continues to apply as a preselection for possible subsequent opening of the Insert function. If this is started, the new part is attached directly to the previously selected anchor without having to re-select it.

## 4.2 INSERT

If this mode is activated, depending on the part to be inserted, in general the program automatically switches to one of the easy to use modes: "Quick Insert", "Insert on Groove" or "Insert Area". However, exotic insert options could possibly not be fully covered by these modes, which is why it is always possible to use the **"Switch to detailed insert mode"** button in the Main window to return to the basic Insert mode described here. 

In this mode the part displayed in the Preview window can be inserted in the assembly. While the insert anchor is already selected in the Preview, **the attachment anchor on which it is to be placed should be selected in the Main window**. If you move the mouse pointer across the screen the parts are automatically selected (marked) and their anchors are displayed. As in the Preview, parts selected here are **displayed transparently**, if you keep the **Shift key pressed** while moving the mouse. If you pause on an anchor for a short time with the mouse pointer, the transparent Preview changes from violet to orange and is rotated in the same way as the part will probably be inserted. The **Rotation Suggestion** is produced by the fact that in general anchors are installed in the opposite direction, i.e. are **installed tip to tip**. **Click the anchor to confirm the insert position**.

You can then **define the rotation**. Following a **direct input** in the Control window, this is **also possible in 90° steps with the mouse**. You can select all directions along the main axes, either of the global or of the local coordinate system (local with regard to the part to which you attach the part). **You can switch between global and local by pressing the Space bar**. However, you cannot use the mouse position to define every possible rotation. The remaining degrees of freedom are covered in the form of a **rotation about the part's own axis with the help of the Tab key**. Alternatively, the **Cursor keys and Page Up/Down can be used to rotate in previously defined (in the Control window) steps**.

In addition, it is also possible to add an **offset** to the **Insert Position** by entering the relevant values in the Control window. The offset can also be defined **with the help of the mouse** if you use the **mouse control** (with the Backspace key or in the Control window) to **switch to "Offset"**.

Several parts, e.g. profiles, also have a **variable length**, which can also be defined during the Insert action. This is possible using the **mouse position** (in an increment defined in the Control window) as well as by **entering the numerical value in the Control window**. The numerical value can also be **entered directly in the Main window (Backspace is then used to Reset)**. Another option is to **automatically adjust the length to a target point**. To do this, the mouse pointer must merely be **positioned above the relevant target anchor**. If the rotation is not in the required direction it is also possible to **"fix" the current mouse rotation by keeping the Ctrl key pressed**. In this case, only the length changes, but not the orientation. In the Control window you can deactivate snapping of target points, if it interferes with the current insertion.

In this mode it is also possible to attach a part, not to an existing part, but freely on the floor (the x-y plane) by clicking in the Main window, outside the snap range of anchors, when selecting the

attachment anchor. This option is illustrated by the fact that if the mouse pointer is paused the Preview changes from violet to yellow and is rotated / positioned accordingly.

Depending on the installed part, the behaviour may differ from that described here so that, for example, rotation is not possible. These are consequences of the parts logic (see Chapter 10.6), which can be disabled if necessary using the corresponding button in the Main window.

### 4.3 Quick Insert

This mode is an alternative to the normal Insert mode, in which logic definitions are used to attempt to limit the selection to the most sensible insert options. An important difference here is also the fact that it is not necessary to select a useful insert anchor in the Preview window first. Instead, only the attachment anchor in the assembly is selected. Based on this selection a suitable anchor is then automatically defined in the part to be inserted. If there are several possible anchors, it is possible to switch through these until the required one is active. To do this, the mouse control must be switched to "Change Anchor" mode (use button in the Main window or press the Backspace key) and then the mouse cursor moved to the left or right. The change is illustrated by the fact that only the relevant anchor can be seen in the Preview window. In the "Rotate around anchor" mouse mode it is also possible to rotate the part to be inserted in the attachment point about its Z axis (also done by moving the mouse pointer to the left or right). Regardless of the mouse control, the Space bar can also be used to change the anchor or the Tab key can be used to rotate it. Here it must be noted that only the rotation about the z axis can be changed while the remaining rotation is defined automatically. The basic principle of installing the anchors tip to tip is possibly not pursued further here, if another rotation makes more sense.

If the required insertion has been confirmed with a click, the length of the part can then be defined, providing it is variable. If the attachment anchor selected at the beginning is a line anchor, in a following step it is possible to define the insert position on it. As with normal insert, numerical values for length and position can also be entered here in the Control window. There it is also possible, as usual, to activate snapping of anchors (target points) for fast definition of the values using the mouse. In this case, snapping to anchors can be used particularly effectively, in order to build several parts at the same height in different grooves.

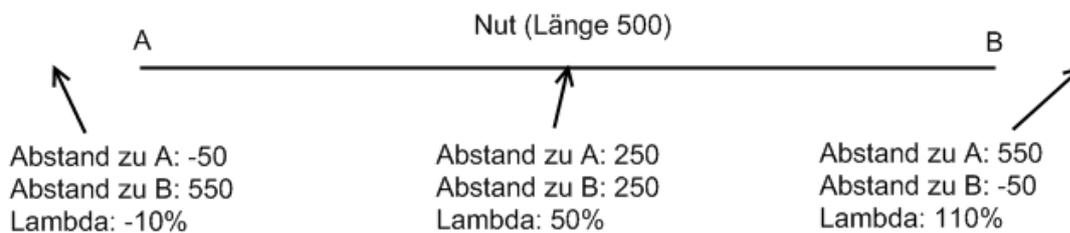
Quick Insert mode is particularly suitable for inserting parts with many different anchors, especially profiles. However, the insert variants are limited to the most important, so that the Insert mode described in detail above must be used for more unusual variants (to be opened via the corresponding button in the Main window).



### 4.4 Insert on groove

Several parts with variable length, e.g. seals are almost always installed in a groove. This can easily be done using the normal Insert mode, however, it is made even easier with the help of this special mode. Similar to Quick Insert, in this case, instead of selecting an insert anchor in the

Preview you initially only select the required groove. You then select the start and end point on the groove for the part to be inserted. It is also possible to choose points on the extension of the groove. The positions can also be given in the Control window by entering the distance to the end points of the groove or as a percentage on the groove. Negative distance values to the nearest end point apply to positions on the extension of the groove, or lambda values  $<0\%$  or  $>100\%$ . Example:



When selecting the end point, instead of defining the position you can define the required length of the inserted part instead (orientation as last indicated with the mouse).

After defining the start and end point, in a final step it is also possible to choose the insertion direction, which can be decisive for parts that do not have a symmetrical cross-section. Here it must be noted that in this mode you cannot define any rotations, as these automatically result from the insertion along the groove and the insertion direction.

#### 4.5 INSERT AREA PART

**Area parts**, i.e. parts whose length and width (and partly the thickness too) are variable can also be **inserted using the detailed Insert mode**; in which case the **length and width can be selected via the Control window**. However, it is more useful to use this **special Insert mode**, in which the area is defined by spanned via 3 clicks. In general 3 corner points of the area are selected, but the restrictions apply here automatically. The first click must be on an anchor in the corner of a profile or at the end of a groove. The two following clicks are used to define the length, width and orientation. Only rotations in  $90^\circ$  increments are possible to align the first anchor. It is also possible to snap to anchors to define the length and width. Both are similar to definition of the rotation and length using the mouse in simple Insert mode. Here it is also possible to "fix" the rotation by keeping the Ctrl key pressed and only change the distance. The mouse control can be switched off using the Backspace key in order to select precise positions on the line anchors.

After choosing the third point it is still possible to **edit the edges of the area** by gripping them near the Letter marking with the mouse and moving them as required (in which case, the program automatically jumps to anchors near the mouse pointer, if it was not deactivated beforehand in the control). Alternatively, you can also enter numerical values in the Control window **to move inwards or outwards**. In this case the edges are individually identified and addressed via letters and colour markings, but you can also enter a value, which is added to each

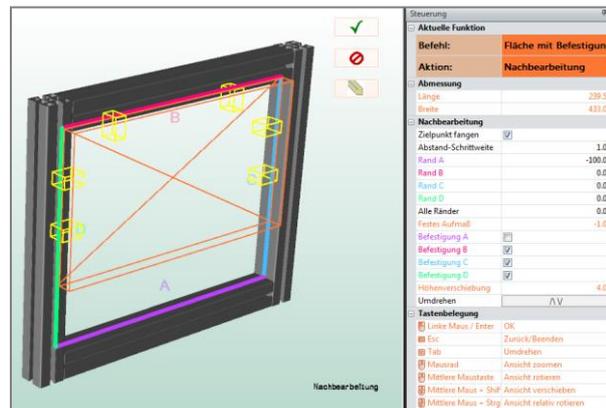
of the 4 edges. It is also possible to enter an **Offset in the insert height**, and to **rotate the plate** (the thickness then stretches or extends in the opposite direction).

Note: The distance between the first and second point equals the length and the distance between the second and third point is the width of the area part. This may have effects on the minimum/maximum sizes and the output of machining plans.

### INSERT AREA PART WITH FASTENERS

In addition to the pure insertion of an area part, this mode also offers an integrated wizard, which additionally inserts the parts required for fixing on the frame. To do this, the clear dimension of the frame must be measured for the 3 click placing of the area part.

After defining the base area in this way, various insertion scenarios are offered for you to choose from, which are possible for the selected area part. Here it also makes a difference whether you select a point in the profile corner as the start point (supported insertion) or at the end of the groove (internal insertion). After you have decided which scenario to use, you may be asked on which grooves the fasteners are to be positioned (only for supported insertion).



This selection is ended by pressing the "Next" button or by clicking the background in the Main window which there are no anchors.

In the final step a preview of the area part is displayed with its fasteners. The dimensions and height offsets predefined by the fasteners have already been applied to the area. It is now still possible to adjust the edges by dragging them with the mouse or by entering relevant values in the Control window. There you can also switch off the fastener (and any associated dimension) for individual edges of the area or check the distance of fasteners. You can also rotate the area together with the fasteners.

It must be noted that when you adjust the edges in the last step, in case of internal insertion the fasteners are moved too, but in the case of supported insertion they remain on the selected grooves.

Note: On positioning fasteners, necessary machining (drillholes) are attached to the area part. These can then also be seen in a machining plan (see Chapter 10.3).

## 4.6 DELETE

This function is used to remove already positioned instances. If **one or several parts are already selected** (marked) when the mode is activated, these are removed and the **mode is**

**then ended automatically.** However, if **no part is selected** on activation the mode remains **active until it is cancelled.** Parts are then removed by simply clicking them.

In addition, there is a separate Delete mode especially for notes and dimensions.

#### 4.7 REPLACE

Replace mode enables one or several parts that have already been installed to be replaced by other articles. The **positioning and rotation are retained.** To this end, in the Main window **the parts to be replaced are selected first** (multiple selection possible in the same way as for "Select"). Then (as long as the mode remains active), **whenever a new article is selected in the Library list, these selected parts are replaced by the new article.**

#### 4.8 COPY

This can be used to **duplicate one or several selected parts.** A new position is then defined for the new inserted copy. The behaviour is **identical to the Move mode described below** (see 5.1) with the only difference that to create the copy you do **not have to press the Ctrl key.** The same effect can also be achieved without this mode by copying the parts onto the internal clipboard with **Ctrl C** (or move them with **Ctrl X**) and then pasting them with **Ctrl V** (alternatively using the corresponding menu entries).

## 5 EDITING MODES

### 5.1 MOVE

This mode is used to move one or several parts already placed freely in the space without changing their rotation. If not parts to be moved have been selected yet, the command begins with a "Select Part" action. The **mouse can then be used to define an axis and distance**. **Direct entry of a numerical value in the Main window** enables precise offsetting **on the last axis selected in this way** (mouse is deactivated). Alternatively, it is possible to changeover the mouse control in the Control window, so that the movement is made **on a specific axis or plane** (direct entry of the distance is then not possible). The **increment** of the mouse offset can be set in the Control window.

The Space bar allows you to switch between the **global coordinate system** and the **coordinate system of the part to be moved** (in case of multiple selection the reference part applies, which is shown in a different colour).

As an alternative to mouse control, it is also again possible to **enter the numerical values in the Control window**. To do this you can use the Backspace key to deactivate the mouse control (unless you are in the middle of making a direct entry, in which case the key is used to undo the entry). The **Cursor keys** and the **Page Up /Down keys** also enable you to **move using the keyboard**. If, while confirming the move by clicking the mouse or Enter key, you also **press the Ctrl key**, the part is **copied instead of moved** (see Copy mode 4.8)

### 5.2 MOVE VIA ANCHOR

While the Move mode described above works independently of anchors, this alternative mode concentrates on their use as an orientation aid. After the parts to be moved have been selected, this command begins by **selecting a start point** followed by the **selection of an end point**. Both points are any anchors in the assembly where, if line anchors are selected, the precise position can be defined with the help of the Control window. The selected parts are then moved so that the start and end point are congruent, i.e. **along the vector that connects the two points**. None of the anchors have to belong to the moved parts, you can therefore "pick off" the vector anywhere. You can also use the Control window to block individual coordinate axes, so that no movement or offset takes place on this axis (only along the unblocked axes). If, while selecting the end point, you also **press and keep pressed the Ctrl key**, the part is **copied instead of moved**.

### 5.3 ROTATE

You can use this command to subsequently rotate an already positioned part (or even several parts). After selecting the part, you first select the **centre of rotation in the form of any anchor** in the scene. It is also possible to switch off the mouse control and instead **to enter any world position in the Control window**. The next action is used to **select the axis of rotation**. Here the **Space bar** enables you to switch **between the axes of the global system and those of the local coordinate system**.

After the point and axis have been defined, **the mouse can be used to define the rotation**. If the mouse cursor is moved on an anchor of the assembly, the rotation is based on this anchor. The Tab key can then also be used to rotate in 90° increments about the selected axis. In this way it is possible to select which of the axes of the part to be installed is to be aligned with the anchor. The snapping of target anchors can also be disabled in the Control window.

As usual the Space bar continues to switch between global and local, while the **Cursor and Page Up/Down keys** can be used **to rotate about all 3 axes**. The Control window again enables **direct input of numerical values** and subsequent changing of the axes for the mouse rotation as well as the **increment**. If, while confirming, you also **press and keep pressed the Ctrl key**, the part is **not only rotated but is also copied**.

### 5.4 EDIT LENGTH (SCALE)

In the case of the parts for which this is allowed, this function can be used to subsequently change the length (possibly the width and thickness too). To do this, first select the part concerned and then **define the fixed point**. This is an anchor of the selected part, which is to remain in its position when the length is changed. The following **lengthening or shortening takes place accordingly at the opposite end**. The change can be defined **with the help of the mouse** or by using the **"Page Up /Down" keys** or by **entering values in the Control window**. Also, as with inserting parts, it is also possible to **enter the length directly in the Main window** (Backspace resets the value). Also analogous to Insert, it is possible to **automatically adjust the length up to an anchor**. This is in turn done by pausing with the mouse pointer on the anchor concerned.

In case of **area parts** with variable length and width, with the mouse control the area is **stretched between the fixed point and the mouse position** (as the orientation is already defined, 2 points are sufficient to define the area). Here too, the program automatically jumps to anchor points. Direct input of values (in the Main window) is not possible for area parts.

### 5.5 RESIZE

This mode is an **alternative** to the "Edit Length" (Scale) function described above. Also, like there, here an already positioned part with variable length and possibly width / thickness is **post-processed**. However, this mode follows a different approach, which **is not based on**

**selecting a fixed point.** Instead, directly after selecting the part to be edited, it is possible to **move the ends or the edges by means of Drag & Drop (important: you must click the centre of the side marked in colour, not in the Preview box).** Alternatively, the modifications can **also be entered in the Control window** (analogous to Insert mode for area parts). There it is also possible to define **the increment for the mouse control** and to define whether or not this should **jump to anchors**. In the case of area parts, the part can also be **rotated** there (thickness then stretches in the opposite direction) and if applicable enter the **thickness**.

It is also possible to apply this mode to a multiple selection of parts. However, only the parts which can be scaled parallel to the reference part (identifiable by their colour) can be edited. Regardless of the position of the parts, the same length is added or deducted at the same end of each part.

## 5.6 DRAG & DROP

This function can be used to move already positioned parts by means of **Drag & Drop**. To do this, the part to be moved **is picked / clicked at its anchor and with the mouse key pressed it is dragged to another anchor of the assembly**. When the button is released the part is **inserted in the new position with its old orientation (and if applicable the same length)**. Before finally confirming the action it is still **possible to adjust the orientation and length** if required. The behaviour is identical to placing a new part with detailed Insert mode. If, while releasing the mouse button, you also **press and keep pressed the Ctrl key**, the part is **copied instead of moved**.

## 6 NOTES, POSITION NUMBERS AND DIMENSIONS

### 6.1 MEASURE

This function can be used to measure the distance between two anchors in the assembly. To do this, the **start point is selected first**. Then **move the mouse over the other anchors** to display the distance from the start point to the relevant anchor. In the case of **line anchors**, it is again possible to define the **precise position** of the start and end point **with the help of the Control window**. If an end point is selected the measured distance is copied onto the clipboard and the measuring procedure is restarted

### 6.2 DIMENSIONS

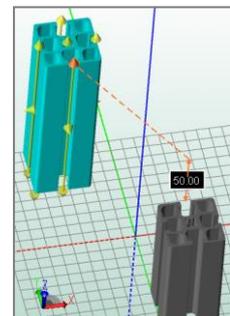
While "Measure" mode is merely used to check distances, this mode generates **dimensional information, which remains in the 3D view of the Main window**. In order to position a new dimensions, you must first **click the start point and then the target point** (anchor). You can use the Control window to select whether the geometry of the parts is clicked directly or whether the anchors are used. A special feature arises if a **line anchor is clicked** and the mode is activated for **automatic dimensioning**. In this case the line anchors cannot be used as end points, but if they are selected as the start point they automatically generate a dimension which accepts the start and end point of the anchor. **Another click sets the dimension text**.

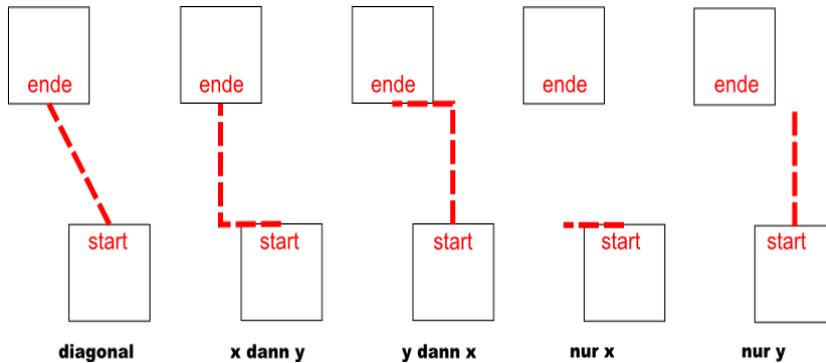
Whenever this mode is active, already positioned dimensions can also be **moved** again by **picking and dragging** them. As in the last step of inserting, the **text is moved but the measured length is not changed**. The dimension is moved, but its ends are connected to the measured points by additional lines.

If a dimension has been selected by clicking on it, the **text can be changed using the keyboard** or it can be **deleted using the "Del" key**. Selecting the dimension also causes the **colour and size** to be set as in the Control window. Other setting options only affect dimensions generated in the future. This concerns both the **number of places after the decimal point** and **restriction to axes or planes**.

If a dimension **is limited to an axis or plane, only the distance along this axis or plane is measured**. **Help lines** (extension lines) can be attached to clearly illustrate the distance that has been measured and to connect the clicked end points. It is possible to choose whether the points are directly connected with each other ("diagonal") or whether the "help lines" are to be broken down into components parallel to the axes. In the case of the latter variant, a decision is also made whether only one of the components is to be displayed and the order in which the axes are to be worked through.

Here is an example of the help lines for a dimension on the Z axis, viewed from above:





Limiting to axes and planes also **affects the moving or offsetting of dimensions**. If such a restriction is active, the movement does not take place on the screen but on the plane corresponding to the restriction. Where this plane is viewed with too small an angle ( $<30^\circ$ ), the anchor is adjusted to prevent moves to extreme distances.

Apart from limiting to axes and planes, it is also possible to **limit orthogonally**. In this mode, the **measurement and movement is always made along one of the 3 axes**. In the case of measuring, **the axis along which the distance between the start and end point is the largest is always chosen**, while it is then possible to move along the two other axes.

In the Control window, the "Update All" button can be used to transfer to current setting for size and colour to all existing dimensions.

The "View" menu () can be used to temporarily hide all dimensions without deleting them. They can be deleted not only during the editing but also in the separate Delete mode for notes and dimensions

### 6.3 POSITION (ITEM) NUMBERS / NOTES

You can use this mode to attach **notes to already installed parts**, which can then be seen in the Main window. A note is also inserted by clicking on the part in the place where the note is to be attached. The text can then be **freely positioned within the 3D space**, while it always remains **connected with the original click position by a line**. The movement takes place on the screen level, the view therefore plays a decisive role for the position. **With another click, the text is placed in the mouse position**. Notes that have already been placed can be **moved again** at any time, provided Note mode is still active, by **picking and dragging the text with the mouse**. Click another note to select it; its **text can then be changed using the keyboard**. Press the **"Del" key to delete** a selected note. The **size and colour** of future notes can be set in the Control window. It is also possible to select the shape of the surround. These settings are applied to existing notes when they are clicked. However, the "Update All" button can be used to quickly transfer them to all notes.

The default **text** selected for new notes **corresponds to the current position (item) number of the corresponding part in the bill of quantities**. If a position number is to be assigned to all

parts, it is not necessary to create a note for each individual part. The control window provides several **algorithms**, which **automatically insert an appropriate note for each part that can be currently seen** and position the text in the space. A **distance factor** can be given, which affects the distance of the text from the part or more precisely the **length of the connecting line**. It is also possible to choose whether the middle point (centre) of the scene or the average of the centres of the individual parts (called the centre of gravity here) is used as the **origin of the algorithms**. Another option "visible only" defines that only parts whose centre **is not concealed by another part** are assigned position numbers.

The various algorithms used are briefly explained in the following:

- **Sphere:** From the centre of each part, the text is moved away from the origin so that the distance from the origin is the same for each text, i.e. all are located on the surface of a sphere.
- **Explode:** From the centre of each part, the text is moved away from the origin and the further it is moved the greater the distance of the centre to the origin.
- **Expand:** The common feature of all 4 variants of this algorithm is that the length of the connecting line is the same for each note inserted. Unlike Sphere and Explode, the movement is no longer straight from the origin, but depending on the variant is adjusted by different amounts:
  - **1:** From the origin, however with a correction factor, which takes into account the different extent of expansion of the scene along the 3 dimensions
  - **2:** From the origin, however each dimension is cubic-attenuated, so that the direction is no longer along the axes
  - **3:** From the origin, however adjusted to 45° angle relative to the axes
  - **4:** From the origin, however adjusted to the axes

The notes positioned by these algorithms can be subsequently adjusted, moved and deleted just like any other note. On the other hand, it must also be noted that after they have been inserted, they are normal notes whose contents **are not automatically adjusted** if changes are made in the bill of materials which affect the position (item) numbers. To update the note to the current status of the bill of materials, you can either press the Pos1 key (affects the currently selected note) or the "Refresh" button in the Control window (affects all automatically inserted notes).

Instead of the position (item) number, other information can be inserted from the bill of material. To do this, the required information is selected in the Control window (apart from the position (item) number, it is also possible to select the Article No., Name/Description, Dimensions and Size). As, if a multiple selection is made, the different entries are placed consecutively, here it is also possible to define a text that is used as a separator.

The "View" menu () can be used to temporarily hide all notes without deleting them. They can be deleted not only during the editing but also in the separate Delete mode for notes and dimensions.

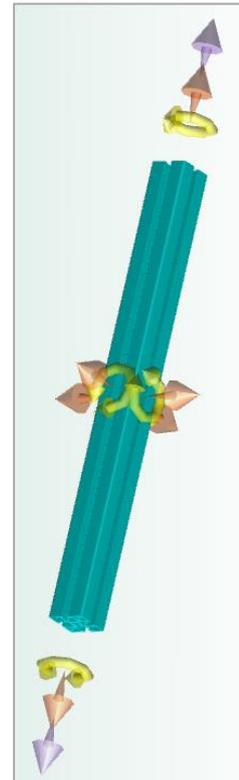
## 7 SIMPLE INTERACTION VIA HANDLES

### 7.1 EDITING SELECTED PARTS

If already installed parts are to be edited, depending on the type of editing (move, rotate, etc.), this can be done in great detail using the respective matching editing mode described above. However, there is an alternative that allows quick editing without having to change mode. To use this, in "Select" mode (if the corresponding option has been activated in the Control window), handles are shown on the selected part, which can be dragged by pressing the mouse button.

The colour of the handles symbolises the editing to be carried out as follows:

- **Yellow – Rotation:** The selected parts are rotated about the displayed axis from the handle and the centre of rotation corresponds to the centre of the handle.
- **Orange – Offset:** The selected parts are moved along the axis displayed by the handle.
- **Violet – Resize:** The size of the selected parts, if possible, is changed in the direction represented by the handle, and the same length is added to or deducted from each part.



All these edit functions are also possible if several parts are selected. However, in this case, it can be decisive which part is selected as the primary part. This part, also identified by the marking colour, contains the handles and acts as the reference for the editing of all the selected parts. Therefore, depending on the primary part, the axes and centres of rotation available for selection can also change. If a multiple selection has been made (by holding the Control button and /or dragging a frame), the part nearest the mouse pointer becomes the reference part. This change in reference part can be prevented if necessary by pressing the Ctrl key and keeping it pressed, in order to reach the required handles with the mouse pointer.

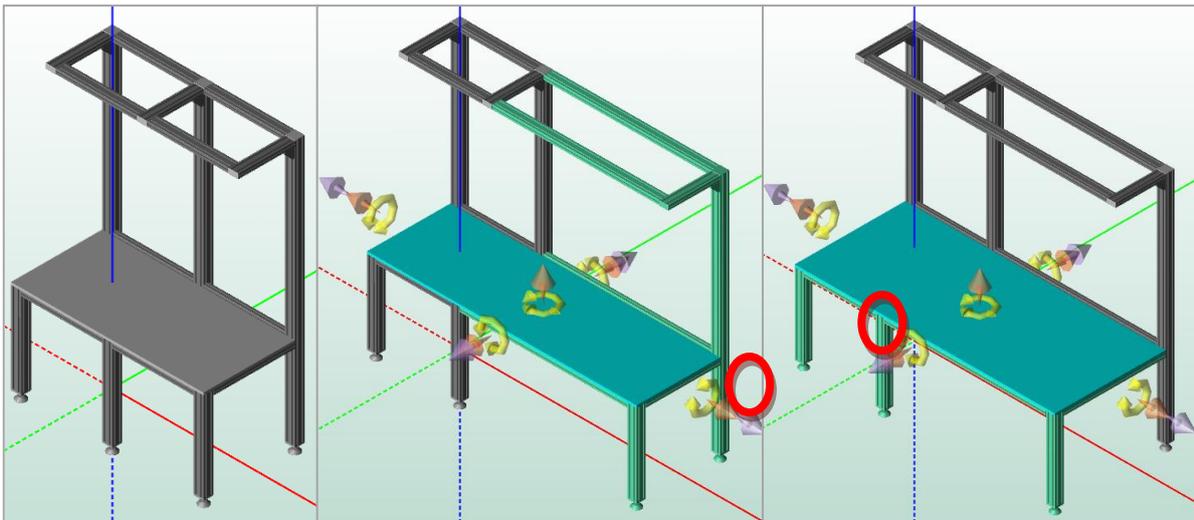
As with the special editing modes, a copy can also be made when moving and rotating by means of the handle by keeping the Control button pressed. In addition, in the Control window (or by using the "m" and "s" keys) it is possible to define whether or not anchors are to be used as the target for a move or size change. Here it is also possible to select the increment for the editing. Another option activates automatic hiding of handles if they are too close to each other due to the viewing angle and zoom.

The handles cannot only be dragged, but also allow the required numerical values for rotation, movement (offset) and size change to be defined directly. To do this, the value must be entered

using the keyboard while the mouse pointer pauses over the relevant handle. If the entry has been started, the Enter key can be used to make the change or it can be cancelled by pressing the "Esc" key. Alternatively, buttons are also available in the Main window, with which you can confirm or cancel.

## 7.2 COMPLEX EDITING THROUGH SIZE CHANGES

In the case of size changes, the handles can be used to achieve effects not possible even with the special modes (in a single step). Here it is decisive to note that the size is only changed for the parts that allow a change in the chosen direction. Unlike "Resize" mode, however, the other parts are not ignored here but are moved according to the size change. By skilled selection of the marked parts it is possible to make complex changes in a single step. The following example of a workbench is used to illustrate this:



Here the width of the workbench was increased first and then its depth. This was done with a single interaction with the selected (violet) handle for size change. It should be noted that both the tabletop and the profiles which were scalable in the chosen direction were lengthened, while unchangeable parts (such as feet and cover caps) as well as profiles located perpendicular to the direction were all moved by the same amount. For each of the two actions, other parts had to be appropriately selected. In a comparable way it would also have been possible, e.g. to change the height or length of the cantilever in one step.

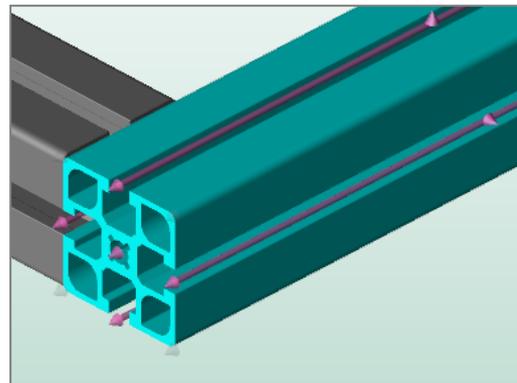
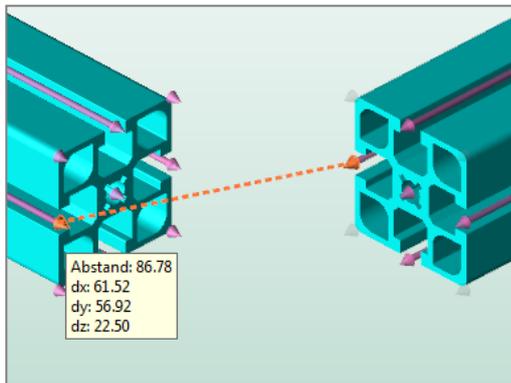
## 7.3 INTERACTION WITH ANCHORS

You can use the Control window or a corresponding button in the Main window to switch off the display of the handles. The anchors of the selected part are then displayed instead. In this case, however, the anchors are not displayed in yellow, but in a violet colour and provide special use options.

With a single click of the anchor (with the left-hand mouse button), the part currently selected in the library is installed in this position by means of the Quick Insert function. In this way, parts can be installed quickly without having to change to an Insert mode.

If you right-click an anchor, instead of the usual context menu, in this case the program searches the library for all parts which could be logically installed in this position (see Chapter 10.6 on Parts Logic later in the document).

Apart from clicking the anchor, here it is also possible to pick the anchor and, with the mouse button pressed, to drag it to another anchor in the assembly. In this case the selected parts are moved so that the picked anchor and the target anchor are congruent. To access concealed anchors, you can keep the Shift key pressed to display the current part transparently.



## 8 MORE ABOUT THE WINDOWS

### 8.1 SEARCH IN THE LIBRARY

The toolbar of the Library list allows a **text search** optionally in the article numbers, descriptions and/or dimensional information of the parts. The search result acts as a filter. The active filtering can be identified by the marking in the toolbar (where the number of parts found is also displayed) and the background colour of the list. It can also be **removed**.



It is possible to combine several search terms with the “+” symbol. The articles in which all the terms linked in this way occur are then displayed. In addition, in general, the “.”, “/” and “\_” symbols are usually ignored to make the search in article numbers quicker. To prevent this, the search term must be enclosed in inverted commas. E.g.: Profile+“45” searches for profiles in which 45 occurs (but not 4/5 or 4.5).

If you enter a search enquiry, the product group marking in the library tree switches to the main nodes to display that the search will be made in the entire library. It is also possible to limit the search to the product group presently active. To do this, before starting the search, the corresponding function must be activated in the toolbar , which prevents automatic cancellation of the group selection. Further, a search filter is cancelled as soon as the group in the tree is changed. This automatic behaviour can also be prevented using the corresponding function to block the filter .

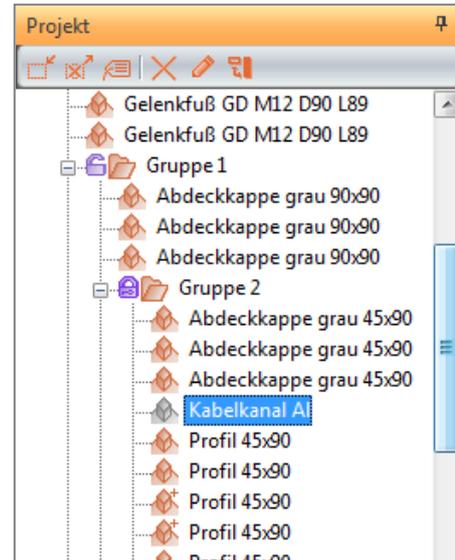
The last column in the table enables a marking to be set for each product and therefore to define it as a favourite. You can use the window's toolbar  to activate a filter, which only displays the parts marked in this way. This filter corresponds to a search for marked parts and behaves analogous to a search filter.

Another search function is based on the logic information, which defines whether or not parts can be installed together. The search for suitable parts  browses the library for all parts which, according to the parts logic (see Chapter 10.6) can be attached to the currently selected part. This search is also like the other search filters. The same also applies to the search for parts that can be installed in a specific position, which is initiated using the "Search for fitting parts" mode or by right-clicking an anchor in "Select" mode (see Chapter 4.1.3 and 7.3).

## 8.2 ASSEMBLY MANAGEMENT IN THE PROJECT WINDOW

**All part instances installed in the current assembly are listed** in the Project window. The selection of an entry here marks the corresponding part in the Main window and vice versa.

However, the actual purpose of the Project window is to **organise parts in groups in any way required**. If you have used the "Select" mode to mark several parts in the Main window, it is sufficient to press the **"Create Group"** button  in the toolbar of the Project window to create a group with these parts. From now on you can **Select the group entry in the Project window to make a corresponding multiple selection in the Main window**. You can use **Drag & Drop on entries in the tree** to rearrange individual parts and whole groups. For example



- by moving one group into another group - you can **set up a hierarchy**. Multiple selection by selecting a group entry always includes all parts in the group including all sub-groups. Groups that are no longer required can be **disbanded**  by pressing the relevant button. Their contents are then moved into the next higher group within the hierarchy.

It should be noted that with grouping, a part **can only ever be in one group**. If you create a new group and use a part that is already in another group, it is **moved from there into the new group**. Otherwise, when you insert a new part in the Main window, it is possible that automatic grouping takes place. If an attachment is made to a part, which itself is in a group, the new part is **automatically moved into this group** (can be switched off in the options). Further **automatic grouping takes place with Copy & Paste**. If a multiple selection is inserted, the parts concerned are automatically grouped together in a new group. Area parts and fasteners are also grouped if the Insert Wizard for area parts is used (see Chapter 0).

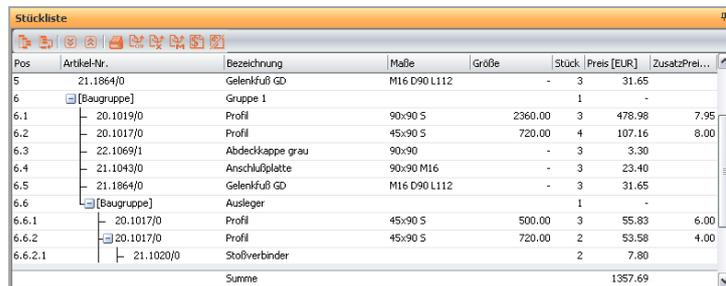
The grouping of parts not only helps to restore suitable multiple selections. The groups can also **appear in the bill of materials** described later in the document. As with individual parts, it is also possible to name groups. This is then also used in the bill of materials as the **name of the group**.

Another function results from the Lock icon  of the groups, which can be clicked to close it or open it. If a part belonging to a group with active lock  is selected in the Main window, all other parts in this group (and its subgroups) are also selected.

### 8.3 THE BILL OF MATERIAL (BOM)

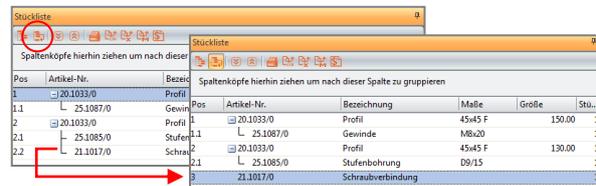
The bill of materials **always** lists the individual parts **currently** used in the assembly. The **article number** and the information taken from the data about the **name** and **dimensions** are also displayed. In the case of parts with changeable length, the **used length** is also given (or the length and width of area parts). Where parts are **used repeatedly in the identical way** (i.e. with the same sub-parts, size details, etc.), instead of a separate entry for each part, the **number of parts is given in the corresponding entry**. If a listed article is a discontinued item, a corresponding warning is displayed . If you **click an entry** the corresponding elements are marked in the Main window.

The bill of materials has a **hierarchical** layout (hierarchy levels are **expanded by clicking the + Symbol**) which can also be identified by the **issue of position (item) numbers**. For example, a **connector assigned to a part is given as sub-items (sub-positions)** in the entry of the part. If configurations are used (see 10.5) the parts from which the configuration is made (if known) are also included as sub-items (sub-positions) In addition, **the group hierarchy defined in the Project window is also displayed in the bill of materials**. The toolbar of the Bill of Materials window can also be used to **place the bill of materials in an alternative mode in which the group hierarchy of the Project window is ignored**. In this mode it is also possible to drag individual columns (represented by their header entry) to a correspondingly marked bar and in this way to group the listing by the entries of the corresponding column. In both modes you can also **change the order of the columns by dragging** or **sort the list by column entries by clicking the header entry**.



Pos	Artikel-Nr.	Bezeichnung	Maße	Größe	Stück	Preis [EUR]	ZusatzPrei...
5	21.1864/0	Gelenkfuß GD	M16 D90 L112		3	31.65	
6	[Baugruppe]	Gruppe 1			1	-	
6.1	20.1019/0	Profil	90x90 5	2360.00	3	478.98	7.95
6.2	20.1017/0	Profil	45x90 5	720.00	4	107.16	8.00
6.3	22.1069/1	Abdeckkappe grau	90x90		3	3.30	
6.4	21.1043/0	Anschlußplatte	90x90 M16		3	23.40	
6.5	21.1864/0	Gelenkfuß GD	M16 D90 L112		3	31.65	
6.6	[Baugruppe]	Ausleger			1	-	
6.6.1	20.1017/0	Profil	45x90 5	500.00	3	55.83	6.00
6.6.2	20.1017/0	Profil	45x90 5	720.00	2	53.58	4.00
6.6.2.1	21.1020/0	Stoßverbinder			2	7.80	
	Summe					1357.69	

The bill of materials also has an alternative display mode in the form of an order list. Here the editing of the corresponding parts is left as it is, but the attached parts, for example screws, are listed at the end of the list as separate items.



Pos	Artikel-Nr.	Bezeichnung	Maße	Größe	Stu...
1	20.1033/0	Profil			
1.1	25.1087/0	Gewin	45x45 F		150.00
2	20.1033/0	Profil			
2.1	25.1085/0	Stufen	M8x20		
2.2	21.1017/0	Schrau	45x45 F		130.00
		2.2.1	25.1085/0	Stufenbohrung	D9/15
		3	21.1017/0	Schraubverbindung	

If a **price list** is available, a **price calculation** is also performed in **additional columns** of the bill of materials and a **total sum** is given. It is also possible to load a price list via the toolbar  or to unload it . When the program is closed and restarted, the last used list is automatically reloaded.

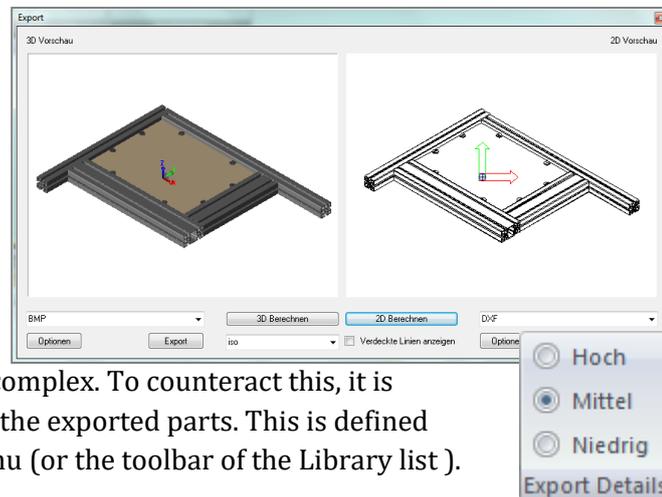
The bill of materials can be output in different file formats or be sent directly to a printer. It is also included in exported PDF documents (see 9.2).

## 9 IMPORT / EXPORT

### 9.1 CAD EXPORT

If you open the "CAD Export" function you can **reconstruct the entire assembly from CAD data and export it in a large number of 2D and 3D formats**. Following completion of the reconstruction, you can use the relevant dialog to select the required 2D and 3D formats and to perform the export. Here it is also possible to calculate and display a preview of the 3D or 2D exports. If a 3D preview has been calculated, the viewing angle in the relevant window is used for the calculation of the 2D derivation, otherwise it is based on a default view which can be selected using the Combobox.

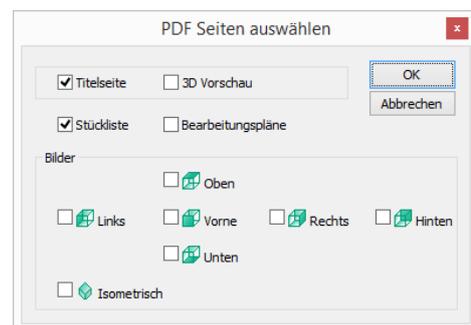
An almost identical function is provided by the "CAD Export (selected)" function. The only difference here is that in this case only the parts currently selected in the Main window are included in the reconstruction.



It should be noted that the geometry of very large assemblies can become very complex. To counteract this, it is possible to adjust the degree of detail of the exported parts. This is defined using a switch in the Import/Export menu (or the toolbar of the Library list).

### 9.2 PDF EXPORT

This function generates a PDF file, whereby it is possible to define beforehand in a dialog, which pages it should include. In addition to the bill of material, it is also possible to select the machining plans of all relevant individual parts (see Chapter 10.3), and a title page. Depending on the selection, the latter contains an ISO view or an interactive 3D preview of the assembly and the project information entered (see 12.2) and the Assembler file of the assembly as an appendix (therefore it is recommended that the assembly be saved before the PDF output).



In addition, pages of the assembly can be inserted, whereby the standard views are available to choose from. The detail window is zoomed so that the entire assembly can be seen. The settings, which define whether position numbers, dimensions, floor grids, coordinate systems, etc. can be seen are also copied over to the Main menu. On the other hand, the background colour and the interactive coordinate system in the bottom left-hand corner of the Main window are not copied into the export.

### 9.3 STEP/SAT IMPORT

You can use this function to import a Step or Sat file and to include it in the library as a new part. After selecting the file to be imported the article number, name and dimensional information are specified. In another dialog it is also possible to define anchors (described in greater detail in Chapter 10.9). The new article is then available in the Library list under the "step/sat" group. Step or Sat files that are no longer required can also be removed from the library in the Cache Overview (open via Library list toolbar or Options).

After the import, parts imported in this way can be installed just like any other parts. If you save an assembly that contains such parts, the corresponding files are automatically appended to the Assembler file. However, it should be noted that for licensing reasons, the Step and Sat parts must be omitted in a CAD export. Equally, they cannot be individually exported via the Library list.

### 9.4 2D DXF IMPORT

You can use this function to open a 2D Dxf file and use it as a floor slab. The display of such a floor slab can be switched on and off using "Display 2D dxf". It is also possible to unload the file. Otherwise a Dxf floor slab behaves similar to a floor slab based on the grid, i.e. it can be moved and rotated, but not copied or removed. The elements contained in the file are not only visually displayed but also have anchors. These are usually line anchors, but point anchors are also assigned, for example to the centres of circles.

### 9.5 THE "INSERT FROM FILE" FUNCTION

Use this function to **insert an Assembler document into another document**. To do this, the function is opened in the current document and the document to be inserted is defined. All parts contained in the document to be inserted are then inserted into the current document in a new group. The **group structure of the inserted parts is retained**. Here you should note that this involves a one-off transfer of the parts from one document into another and not referencing. Subsequent changes to the inserted file are not copied into the files into which it was inserted.

### 9.6 SAVE AS VRML/X3D

The assembly is usually output in 3D formats using the CAD Export function described in Chapter 9.1. Because the assembly has to be reconstructed here using CAD data, the runtime can become very long. The function described here can be used to alternatively output the assembly very quickly as simple area data in VRML and X3D formats. In this case the models used for the display are output directly without using the CAD data.

### 9.7 SNAPSHOT

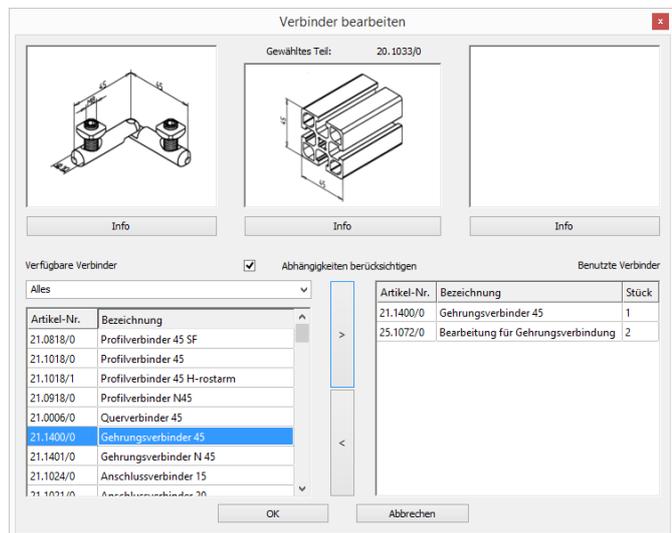
With this function it is possible to output an image of the current assembly as a bmp or jpg file. Here it is possible to choose the resolution of the image, the default view used and whether a perspective or orthogonal view is required. Instead of one of the standard (default) views, it is also possible to choose from several predefined combinations of standard views of an individual image.

## 10 ADVANCED TOPICS

### 10.1 SUB PARTS

It is possible to assign sub-parts to a part in the assembly by dragging the relevant article from the Library list by means of Drag&Drop and dropping it onto the entry of the part in the Project window. There you can recognise parts with attached sub-parts by the small +symbol  inserted in the icon of their entry. The selected sub-parts cannot be seen in the assembly, **however, they do appear with the corresponding part in the bill of materials.**

Sub parts are often connectors or machining operations. The "SubParts Setup" function therefore provides an alternative option for **assigning Sub parts** to one or several selected parts in the assembly. The **Sub parts are inserted or removed here using a separate dialog.** The part to which the Sub parts are assigned is displayed in the middle. Below it, **on the left-hand side you can see a list of all possible sub parts** (filterable by connectors, machining operations and accessories) and on the **right-hand side those that have already been inserted into the part.** You can use the **arrow buttons** between these two lists to **add or remove** the respective selected connector. Whether **possible dependencies are to be taken into account** or not can be switched on and off using the corresponding option. Other sub parts (attached by Drag&Drop) can also be removed (but not added) using this dialog.



Not all parts are intended to have connectors assigned to them. If this is not the case, when the function is opened, the dialog described above is not displayed and instead a variant that only allows sub parts to be displayed and removed is shown.

For an improved overview, in the "Tools" menu you can use the appropriate function  to select all parts in the assembly that have one or several sub parts.

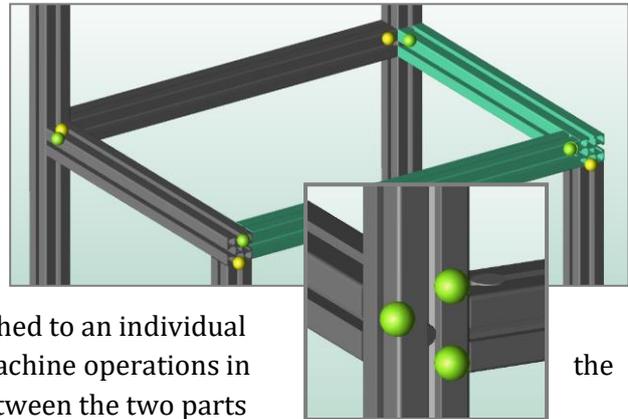
### 10.2 CONNECTIONS

As described above, it is possible to handle connectors by assigning them to the parts to be connected as a sub part. However, here the problem arises that the machined operations necessary for the connector are all assigned to a part when the dependencies are used, although in fact they should have been shared between the two parts involved. The connections represent

an alternative option to distributing connectors, which solves this problem and also provides further advantages.

While parts are assembled in the Assembler, where possible, in the background the program tracks which parts are related where. To this end, connections are created, which have a specific position and link to the two parts connected by them. If one of these parts is edited, the connection is also moved accordingly or if applicable is removed. When parts are edited, new connections can also be created, e.g. if a profile is lengthened by so much that it touches another.

Connections can normally not be seen, but are displayed as spheres as soon as the "Edit connections" function  is activated. If you move the mouse pointer over such a connection, the two connected parts are marked. By clicking a connection it is now possible to assign a connector to it. It may also be necessary to select the correct position. If this has been done the connection is no longer displayed in red but in green. If no possible connector is found the connection disappears. In this way, it is possible to directly trace which connections have already been processed and which have not been. Because the connector was assigned to a connection and was not attached to an individual part as a sub-part, connectors and necessary machine operations in bill of materials can now be suitably divided between the two parts concerned. In addition, the position of the connector can be traced. If activated in the options, the connector can also be partly seen in the assembly.



It is possible that the automatic generation and adjustment of the connections does not correspond to the required result. Therefore, adjustments can be made. For example, while "Edit connection" mode is active you can delete connections by moving the mouse pointer over them and pressing the Remove button. You can also generate new connections. To do this, you must select at least two touching parts and then start the "Add Connection"  function. The program then tries to connect the parts in pairs, whereby the respective possible connectors are offered to choose from.

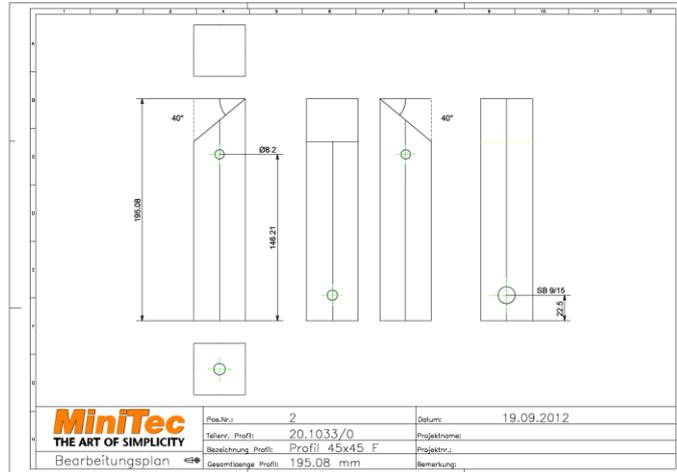
The "Set up all connections" function  enables the same connector to be assigned to many connections without having to select them individually. After selecting a connector, all connections not yet edited (i.e. red ones) are assigned to it (if possible there). Optionally, connections that have not yet been found can also be searched for or those already set can be overwritten.

It should also be noted that the connectors assigned to connections do not count as sub parts. They therefore do not appear in the dialogs described in the relevant chapter, but are displayed in the bill of materials in the same way.

### 10.3 MACHINING PLANS

As described above, the connectors enable the tracking of where on the profiles the connecting elements and any associated machining are to be placed. This information can be used to output a plan for each profile or area part, in which the necessary machining is listed and dimensioned.

The machining plans are output via the “Export machining plan” function . This outputs the plan for a part selected in the assembly optionally as a dxf or pdf file. It is also possible to select several profiles / area parts and to start the function. In this case a target folder is selected instead of a target file, and a separate machining plan for each selected part is saved in the folder as a dxf file.



Alternatively, it is also possible to write the machining plans of the entire assembly in a single PDF file. This is done using the PDF export function described in Chapter 9.2.

### 10.4 FREE MACHINE OPERATIONS

In the machining plans described above the machining operations created by the connections appear automatically. It is also possible to insert additional machining operations manually, which then appears in the plans. “Insert machining” mode is used for this. Here the profile / area part to be machined must be selected first, and then the required machining. This is then placed on the part. The type of machining determines on which anchors it can be placed. In the case of profile drillholes, the precise positioning takes place after placing on a line anchor. The drilling can be placed temporarily with the help of the space bar, so that it can be moved onto different axes, one after the other.

It should be noted that machining operations positioned in this way always appear in the bill of materials and in machining plans, but for performance reasons, are only shown in the Main window if this has been activated in the options.

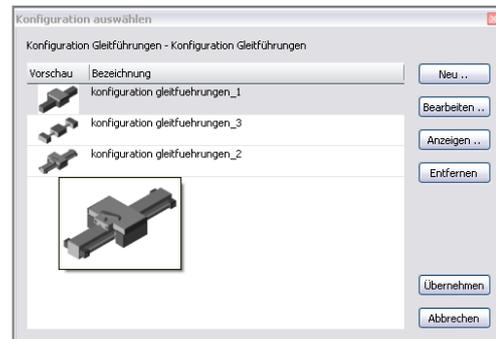
Machining operations can be removed from a part using a button in the Control window of the “Select” mode.

### 10.5 CONFIGURABLE PARTS

While most parts have a fixed geometry or at most their length can be changed, there are several which are **freely configurable**. These can be identified in the library window by the + in the symbol . If such an entry is opened (for technical reasons a product group with only one

entry), a **dialog opens in which already created configurations can be selected and new ones can be created**. If necessary, the dialog can also be opened again later using a button  in the toolbar of the Library list for the part selected there.

The dialog for selecting a configuration **lists all already created configurations stored in the cache together with a preview image**. You can then use the **"View"** button to again display the respective configuration for the entry just selected. If the selected configuration is not used in the current assembly, it can also be **deleted from the cache** here. On the other hand, to create a new configuration, you either use the **"New"** or the **"Edit"** button. In both cases a **window is opened where a new configuration can be created**. The difference is that **in the case of "New" the default configuration is used as the starting point, while in the case of "Edit" the same configuration as for the entry just selected is preset** (despite this, a new configuration is created and the old one is retained).



A 3D preview of the part can be seen in the Configuration window, which is usually updated as soon as a change is made to the configuration. To work more quickly it is also possible to deactivate the automatic update and to perform it manually when required. The preview always shows the lowest degree of detail, but the degree of detail for generating the part can be selected in the window.

Analogous to the "SubParts setup" function, in the "Tools" menu there is also an **"Edit Configuration"** function, which is applied to a selected configurable part. The dialog for **selecting a configuration** then opens where the **current configuration is marked**. It is now possible to **select an already created configuration or to create a new one** in the familiar way (usually using the "Edit" button so that it is based on the current one). If the **dialog is closed with "Apply"**, the **part to be edited is replaced by the selected configuration**.

For some configurable parts it is possible to break down (explode) the installed characteristics into their individual parts. In this case, when selecting a corresponding instance an Explode button is displayed in the Main window. If it is used the configurable part is removed and is replaced by its individual parts. Apart from undoing the explode action directly, it is then no longer possible to re-make a configured whole part out of the individual parts.



## 10.6 PARTS LOGIC

Check **logic restrictions** (if the necessary information is available in the data) **whether insert part and attachment can be installed together with the respective selected anchors**. On the one hand, this is done **on inserting a part**, in that **parts and anchors which cannot be attached cannot be selected**. On the other hand, the **"Search for fitting parts"** function enables you to **search for all parts that can be installed in the selected position**. These are



set as a filter in the Library list (comparable with a text search). There is a similar search in the toolbar of the Library list, where it is possible to filter out parts that fit the selected part. Also, **any anchors that are not allowed are hidden in the Preview window and cannot be selected.**

During the insertion it is possible to temporarily suppress the logic restrictions. This is done directly in the Main window using the corresponding button. In the options there is also a switch, with which the logic restrictions can also be completely switched off. However, in this case you must note that these method of circumventing the restrictions do not work with all functions. For example, the Quick Insert mode is dependent on the logic information and therefore uses its regardless of the settings. However, if the logic is switched off in the options, detailed insert is used as a default, and can also do without this information.

The logic information also enables automatic insertion of cover caps  and assembly brackets  using the corresponding functions in the "Tools" menu. Caps are installed on all unused ends of the selected profiles, whereby, if applicable, it is possible to choose which cap is to be used. In the case of assembly brackets, on the other hand, you must select precisely two profiles. An analysis is then performed of how these parts can be connected with angle brackets. If necessary, the required type of bracket is selected and a query is made for each possible insertion position, whether a bracket is to be inserted is not.

The same logic information is also used in many other places, e.g. for automatic generation of connections. At times steps during insertion are skipped, if it has been defined in the logic that they are irrelevant for the part (e.g. the rotation step for rotationally symmetrical parts).

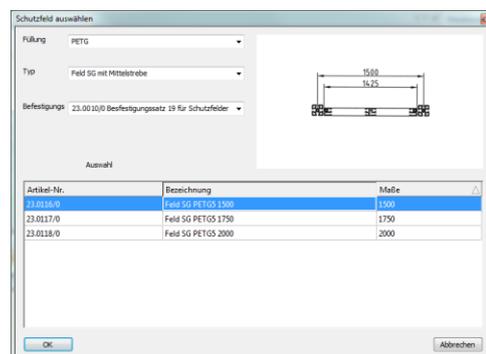
## 10.7 SAFEGUARD GUARDS

The Minitec SafeGuard protection system offers standardised modules for installing guards. The relevant fields and posts can be set up especially fast and easily using the Wizard described here.

Directly after starting the Wizard the user has the choice of whether or not to extend an already assembled post. If they decide not to the Wizard begins with selection and positioning of the first post, otherwise it skips this and goes directly to insertion of the first fields. The Wizard then switches between the insertion of fields and posts until it is ended using the corresponding button in the Main window or using the Esc key.

### *INSERTING FIELDS:*

To insert fields it is first necessary to define which field is to be inserted. In the dialog box used for this it is possible to filter the selection list by material for the filling (infill) and type of field. In addition, the user also defines here which fastening kit is to be used to insert the field. (The fastening kit cannot be seen in



the assembly but is appended to the field as a sub part which then also appears in the bill of materials.)

After selecting the field the user can then position it with the mouse. It is always attached to the nearest anchor of the preceding post. The insertion angle can be adjusted by the increment given in the Control window. However, it is also possible to enter the required angle directly using the keyboard or in the Control window, where the value is given as a deviation from straight insertion. If the mouse pointer is moved far enough from the last positioned post, it is possible to insert several fields of the same type consecutively in a straight line with only one step. The necessary posts are also inserted at the same time. An offset is only necessary where the field to be inserted or the construction direction changes.

Before an insert is confirmed using a left-click in the Main window or pressing the Enter key, several adjustment options are available, which can be accessed using the buttons in the Main window or using keyboard commands. For example, it is possible at any time to change the field to be inserted or the posts to be used for several fields (F/P key). It is also possible to skip the types of positioning of the posts (Tab key) or to rotate the fields, i.e. to swap the inside and outside (Space bar).

If a door is chosen as the field to be inserted, several points of the insert behaviour are modified. For example, only one door is ever inserted simultaneously and it is not possible to modify the insertion angle. In addition, the option for rotating the door is supplemented by a further switch, which defines the side towards which the door opens (Tab key). This is also visually clarified by additional extension lines.

Like with other modes, the Control window can be used to switch off mouse control (Backspace key). In this case, in general only one field is inserted and a button in the Main window (or the Tab key) can be used to define the anchor of the last post to which the attachment is made.

### *INSERTING POSTS*

The insertion of posts functions similar to the insertion of fields, however, with less options. Here the mouse can only be used to affect the insertion angle. Also, a button or the Tab key is used to skip the anchor of the post to be used.

## 10.8 ASSEMBLIES

With the help of assemblies it is possible to define assemblies created in Assembler as new parts which then appear in the library and can be installed just like other parts. It is also possible to dismantle installed assembly parts back into the component parts from which they were assembled. One example of such assemblies are the guards of the Wizard described above. These can be dismantled into their individual parts and then edited in order to obtain individual fields outside the SafeGuard standard.

To create your own assembly, you must first assemble it in an Assembler document, just like with any other assembly. You must then select the muss "Create Assembly 

menu. This function creates an assembly from all the parts currently selected in the document. However, it is still decisive, which anchors the created assembly part should have. In general it does not make sense to apply all the anchors of the individual parts. Therefore, you must first select the anchors of the assembly, which are to appear later in the assembly part. Anchors can be selected and deselected however you wish until the selection is confirmed by pressing the corresponding button in the Main window. You then enter the article number, name and dimensions of the assembly part, as it is to appear in the bill of materials when it is used. After you have finished creating the assembly the parts is then available in the "assemblies" group within the library and can be installed. You should note when you create the assembly the degree of detail selected in the Import/Export menu applies. The assembly part appears accordingly detailed within the program and export.

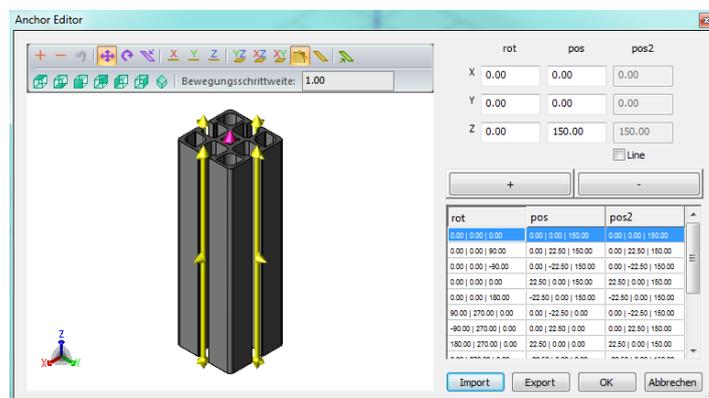
Alternatively, assemblies can also be imported . In this case an assembly is generated from a previously saved Assembler file.

An assembly can be exploded (dismantled) into its individual parts at any time. This is done after selecting the relevant part using the "Explode Assembly " menu function. The function is also available as a button in the Main window if the "Select" mode is active and one or several assembly parts are selected.

## 10.9 ANCHOR EDITOR

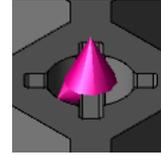
On importing a step or Sat file into ccAssembler the user is faced with the problem that, unlike the delivered article, this initially does not have any anchors (snap points). Yet, as described at the beginning, these are decisive to installed the parts suitably or to attach other parts to these parts. Therefore, directly after the import, a window opens in which anchors can be defined for the part. This dialog, described in the following, then can then be re-opened later to be make changes, if the part concerned is not installed in any currently open document. It is then opened using the "Configurations" button in the toolbar of the library list.

A 3D view of the imported step/sat file can be seen on the left-hand side of the editor. In the same ways as in the Preview and Main window, here the middle mouse button can be used to rotate, move and zoom. If the Shift key is kept pressed the model is displayed transparently. Furthermore anchors already placed here can be seen. These can be selected with a left-click, following which their colour



changes from yellow to red and their values are displayed on the right-hand side of the dialog and can be changed. This direct input of the values is the most precise way possible of placing the anchors. The decisive value here is, of course, the position, whereby in the case of line anchors a second position (start and end point) must also be given. The rotation of the anchor is

given as a third value, to influence the default orientation of the part on insertion. This is described as a Euler angle, in the order: x,y,z. The start orientation (0,0,0) corresponds to the global coordinate system, following which the anchor is then rotated by the given angle value about the X-axis, then the Y-axis and finally the Z-axis to reach the final orientation. In the 3D view, as usual,



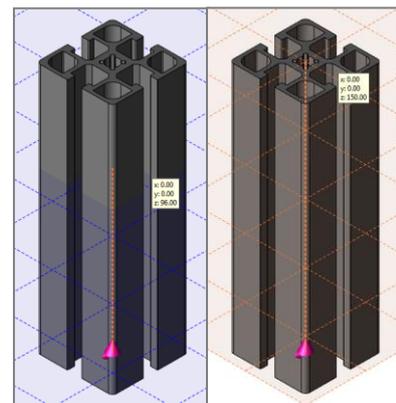
the Z-direction of the anchor is displayed with a large cone and the X-direction with a small cone. Equally, on the right-hand side of the dialogue there is an overview list with the values of all defined anchors. If an anchor in the model is clicked, the corresponding entry in the list is marked, and vice versa. The large "+" button can be used to add a new anchor, the "-" button is used to remove the currently selected anchor.

Direct input of the values is precise, but not very convenient as it requires precise knowledge about the dimensions of the imported part. Therefore, it is also possible to move and align the anchors interactively in the 3D preview. As here the work is on a polygon model without CAD information, the options of automatic orientation on the model are highly limited. For example, it is not possible to move an anchor automatically to the centre of a drillhole. Instead the user must visually orientate it on the model and specify a meaningful movement increment.

After changing to Move mode , the currently marked anchor can be gripped and moved (in the case of line anchors, one of the balls at the two ends). The move is made on one of the 3 main axes of the global coordinate systems or one of its own main planes. This can be selected using the corresponding icons.



Here one of the few options for automatic orientation on the model takes effect. If the option for snapping surfaces  is activated, the model surface under the mouse cursor is used to define a plane (by means of the point concerned and its perpendicular). In addition to the limitation to one of the main axes or planes, if possible, the movement is then limited so that the anchor lies on this surface plane. If the anchor is moved, e.g. on the Z axis and snaps a surface that lies parallel with the x-y plane, the anchor position is defined precisely by the combination of both limitations. Visually, valid snapping can always be identified by the fact that the displayed auxiliary plane is no longer shown in the axis colour but in orange.

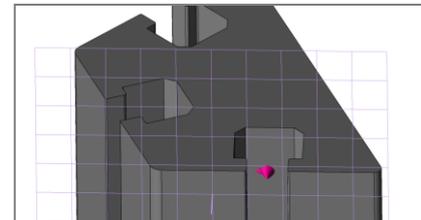
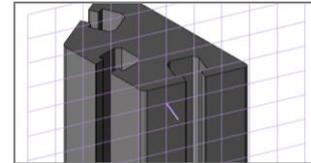


Alternatively, the movement can also be limited only to the surface of the part, if the corresponding limitation is selected . In this case the orientation of the anchor is also set according to the surface perpendiculars.

Just like the position(s) of an anchor, its orientation can also be changed interactively, after Rotate mode  has been selected. Technically speaking, no rotation takes place here, but instead an orientation is chosen, whereby only the orientations along the positive or negative

axes are available. The anchor is first dragged in the required direction with the left-hand mouse button to define the Z-axis of the anchor. The right mouse button can then be used to select the orientation of the X anchor from the remaining 4 directions.

For some parts an anchor should be placed on a surface that is not parallel to a main plane in the space. As this can be difficult using the means described above, a special mode  has been implemented, with which a surface can be selected (in the same way as the already described snapping of surfaces) to define a user-defined plane. Instead of the usual restriction to main axis or plane, a movement can also be limited to this user-defined plane . However, unlike with a main plane, the movement is not only made parallel with the plane. Instead the anchor jumps to the defined plane and also adjusts its orientation accordingly. If the snapping of surfaces is additionally activated, the anchor can then only be moved on the intersection of the two planes (if one exists). In the example shown in the figures, with snapping of the surface of the part, the anchor would only be able to be moved along the corresponding edge.



## 10.10 CACHE OVERVIEW

While most parts in the Assembler are fixed, there are also several scenarios in which variable parts are created, which are then filed locally in a cache. These include characteristics of configurable parts (Chapter 10.5), assemblies (Chapter 10.8) and imported step/sat files. The toolbar of the library list can be used to show an overview of the current contents of the cache . In this dialog it is possible to search for descriptions given by the user on creating the entries. Filtering by configurations/assemblies and imported files also makes it easier to find the searched for files. This can be selected (e.g. to insert the part) or deleted from the cache (as long as the part is not installed in an open file).

## 10.11 ARRANGEMENT (LAYOUT)

The Arrange function allows several copies of one or several selected parts to be generated simultaneously and placed at regular intervals. To do this the coordinate axis on which the copies are to be placed is selected and the number (in addition to the existing original) and the respective spacing defined.

## 11 OPTIONS

The "View" menu or the options menu in the top right-hand corner of the Assembler window can be used to **display** and edit several options **in the Control window**. These should now be briefly described in greater detail here (a description of the entry just selected is also always visible in the bottom of the Control window). The settings are retained when the program is closed and restarted. The option of restoring the original options settings via the Options menu should therefore also be noted (also resets the options of the different modes/actions).

### 11.1 GLOBAL OPTIONS

- **Logic Restrictions** activates the checking of the parts logic described in Chapter 10.6
- **Auto Grouping** defines whether, when parts are inserted, they are automatically inserted into the group of the attachment and whether or not a group is automatically formed on inserting from the clipboard or using the Insert Wizard for area parts.
- **Auto Connector dialog**, if the option is activated, automatically opens the dialog for editing the sub-parts after inserting a part (if relevant for the part)
- **Auto Zoom**, if the option is activated, automatically performs an automatic "Zoom All" after inserting a part
- **Add Parts Separately**, if the option is selected, automatically ends Insert mode after inserting a part
- **Selection Proximity** defines how near an object the mouse must be in order to select it
- **Anchor Size** is a scaling factor for the display of the anchors and handles

#### 11.1.1 VIEW

- **Z Direction Upright** affects the behaviour on rotating the view with the middle mouse button pressed down. If active, the Z axis is always perpendicular and points upwards, i.e. the view cannot be "tilted to the side". This simplifies orientation in space.
- **Reverse zoom** changes the direction in which the mouse has to be turned to zoom in and out.
- **Invert Transmission** inverts the behaviour of the Shift key if it is used to display parts transparently (the display is then transparent as long as the key is not kept pressed)
- **Transparency Mode** defines how the parts preview is displayed, e.g. on inserting or moving. This setting has a large effect on performance
- **Anti Alias** activates anti-aliasing (edge smoothing) for improved display quality

- **Background Top / Bottom** changes the two colours which as colour blending form the background of the Main window

### 11.1.2 RIGHT CLICK BEHAVIOUR

This refers to activation of the alternative control method described below in Chapter 12.1. If applicable, here it is possible to switch between larger and smaller display of the Gesture menu (and therefore the gestures themselves).

### 11.1.3 PRINT

Several options are available here for **printing out the geometry displayed in the Main window** which affect the image to be output:

- **Use Paper Format** adjusts the section of the image to the paper format. If not activated the side ratio of the Main window is used, as can be seen on the screen.
- **Adjust to paper size** adjusts the size of the printout with the same side ratio so that the sheet is filled.
- **Size Factor** is a factor for the size of the printout on the paper. Values from 0.1 to 1 are allowed, where 1 corresponds to the full size.
- **Zoom All** zooms out, with the same viewing angle, so that everything can be seen (corresponds to the "Zoom All" function in the Main window).

## 11.2 DOCUMENT OPTIONS

Here you can make several settings, which concern the currently active document. This includes a perspective display as well as an alternative View mode based on edges instead of filled geometry or the display of edges in addition to areas. An alternative lighting method (Phong) is also available.

The **Display Features** option determines whether or not the machine operations and connectors assigned to the connections are displayed in the assembly or not (see Chapter 10.2). Waiving this display improves performance.

### 11.2.1 GRID

Here it is possible to configure the **auxiliary grid** (see 2.1) and therefore the **floor slab** based on it by giving both the **size of the individual cells** and their **number** (quantity). Alternatively, it is also possible to give the **total size of the grid**, in which case the size of the individual cells is then adjusted accordingly, while their number remains the same. All details can be given **separately for the X and the axis**. Here it is also possible to choose between two variants, how the grid is to be positioned within the global coordinate system: The coordinate origin is either located **centred** in the middle of the grid (in which case the number of cells must be even) or in one of the corners (the grid then extends along the positive coordinate axes).

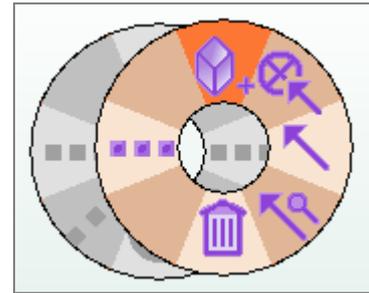
### *11.2.2 3D COORDINATE SYSTEM*

This can be used to replace the **coordinate system** of the Main window represented by lines with an **alternative display whose axis length can be freely selected**. The positive axes then each have the full length and the negative axes 1/10 of the given length.

## 12 MISCELLANEOUS

### 12.1 GESTURE CONTROL

If activated, gesture control replaces the Context menu and therefore the function of the right-hand mouse button. As long as the button is pressed the gesture control shows a **circular menu**, on which there are submenus (represented by "...") and possible commands. The selection is made **not by clicking the icons but by moving the mouse in the corresponding direction**. The advantage of this control option is that a user who has **remembered the arrangement of the symbols** can **select the functions very quickly** without having to search long for them. The selection is made via a **specific movement pattern of the mouse or in other words with a gesture made with the mouse**. For example, it is possible to quickly note that moving the mouse to the right and the left and up opens Insert mode while moving it to the right and then to the left and down deletes a part. By moving to the right and downwards it is possible to reactivate the last used mode. Operation via gestures takes getting used to but is then very effective.



### 12.2 PROJECT INFORMATION

Additional information about the relevant project can be appended in the files in which the Assembler saves an assembly. The input and display of these values, e.g. invoice and delivery address, e-mail, phone number or customer number, are made using a dialog opened using the "Project Information" function. The information entered here is also used on the title page of the PDF output (see Chapter 9.2). The first time a new assembly is saved the dialog for the project information is opened automatically.



### 12.3 GIMMICKS

Several other functions are available via the "View" toolbar, which are used to display the assembly. On the one hand, this is the option of showing a **sectional plane**. Its position and rotation can be influenced by dragging the displayed arrows.



It is also possible to **animate** the displayed scene. A walk through (camera movement) around the scene is available for this, as well as a type of "Explosion" in which all parts can be moved away from the zero point of the global coordinate system.



## 12.4 TIPS

### *12.4.1 SELECTING THE RIGHT ATTACHMENT*

As, depending on the view, parts can often conceal each other, when selecting an anchor it can be difficult to mark the required part in order to see its anchor. Therefore, the system for selecting the marked part is briefly explained here. Depending on the current action, there are two different methods for making the selection.

With the first method, e.g. used for the selection of the attachment anchor in "Insert" mode, the marked part remains active until the mouse pointer leaves it. This is the case even if there is another part in front of it in the current view. If you want to mark the front part you therefore have to move the mouse pointer so that it no longer points to the already marked part. Alternatively, you can also press the Pos1 key to return to the previous action (in this case the part selection) and therefore automatically mark the front part. (By contrast, the Esc key would usually end the mode).

The other method is used if an anchor is selected that does not have to belong to the marked part, for example, if the centre is selected in Rotation mode. Here the program always switches to the front part, even if the mouse pointer is still on the rear part. To prevent this you must keep the Shift key pressed.

### *12.4.2 SELECTING THE RIGHT ROTATION*

For many parts the right insertion situation is sufficiently clear, so that the suggested rotation can usually be accepted. However, if it is necessary to adjust the alignment, due to the combination of mouse movement and pressing the Tab key, achieving the required rotation can be problematic. To simplify this, you should always be aware that when the Tab key is pressed the rotation is always about the axis nearest the position of the mouse pointer. In the part to be inserted, this corresponds to the direction of the selected anchor. To return to the suggested orientation, you must position the mouse pointer on the axis that runs along the anchor on which you attached the part (the anchors are then once again aligned tip to tip).