

Virtual Steel

Version 11.00

Quick Start Guide

Jan 2020

Contents

Introduction.....	4
Workflow.....	4
The User Interface.....	6
Creating the Structure.....	7
Beams.....	7
Basement Objects.....	9
Help Points and Help Lines.....	9
Intersection Points.....	11
Help Lines.....	11
Editing the Structure.....	12
Structural Editor.....	12
Adjustment of Points.....	12
Help Points and Help Lines.....	13
Editing the Details.....	14
Object Properties.....	14
Additional Parts.....	15
Inserting Additional Parts.....	15
Moving Additional Parts.....	16
Copy Additional Parts.....	16
2D-Drawings.....	17
Bill of Materials.....	18
Extras.....	20
Configuration and Settings.....	20
Project Manager.....	20
Project Informations.....	20
Building Grid.....	20
Import and Export.....	21
Groups.....	21
Appendix.....	23
Mouse Hotkeys Overview.....	23

Introduction

Welcome to Virtual-Steel. Virtual-Steel belongs to the group of CAD-Software, which is specialized on a certain theme, in this case **steel constructions**. In contrast to other software products, the only purpose of Virtual-Steel is to **create a virtual model with steel objects**, not with lines, circles, arcs or other drawing elements. The user just needs to act with objects like beams, columns, girders, plates, stiffeners, bolts and hole patterns, just to mention a few. So it is an easy to learn software. Beyond that, Virtual-Steel is a system, which is working on it's own, not a plug-in for another CAD-System. But it was developped to serve as a generator for drawings, which will then be inserted in the CAD-System, which is already running in the planning office. The advantage for users is, that they are familiar with their CAD-System and have no difficulties to complete the drawings.

A very important characteristic is the following:

All steel beams are using a definition line, which is not necessarily the neutral axis, but is used for applying connections. Only if the lines of two beams are in contact by using the same endpoint or intersect each other, connections are possible between these beams. This concept is well known in statics, where cross sections are represented by lines. So please remember:

Virtual-Steel is using a kind of a statics system to create connections.

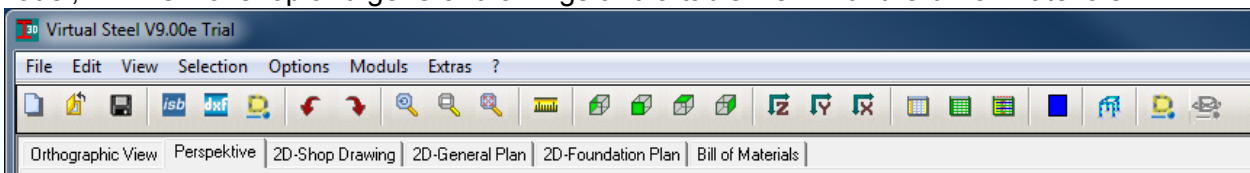
So it is already important at the beginning of a project, to choose a correct structural system. The best way to understand ist, is to work through the tutorial, which is inclosed in the system package and can be opened in the help menu of Virtual-Steel.

More details can be found in the chapter Creating the Structure:

Workflow

In this chapter, an overview of the workflow is shown. It is just for the reason, to show the ideal way of the construction process, as the user can return to any single step later when working on the project.

Below the well known menu bar and toolbar, a line is located, containing different tabs, which from left to right represent the workflow. Activating an other tab will change the view in the main window. Although there are six tabs, there are strictly speaking just three different views: 3D-View of the model, 2D-View of shop and general drawings and a table view with the bill of materials.



In the following graphic the workflow is shown in an overview. It is kept simple, as steps like creating a building grid, switches for dimensions or other settings are not included.

1. Create the structure / model

2. Create the connections

3. Edit the details

4. Numbering and editing the BoM

5. Export/print 2D drawings and BoM

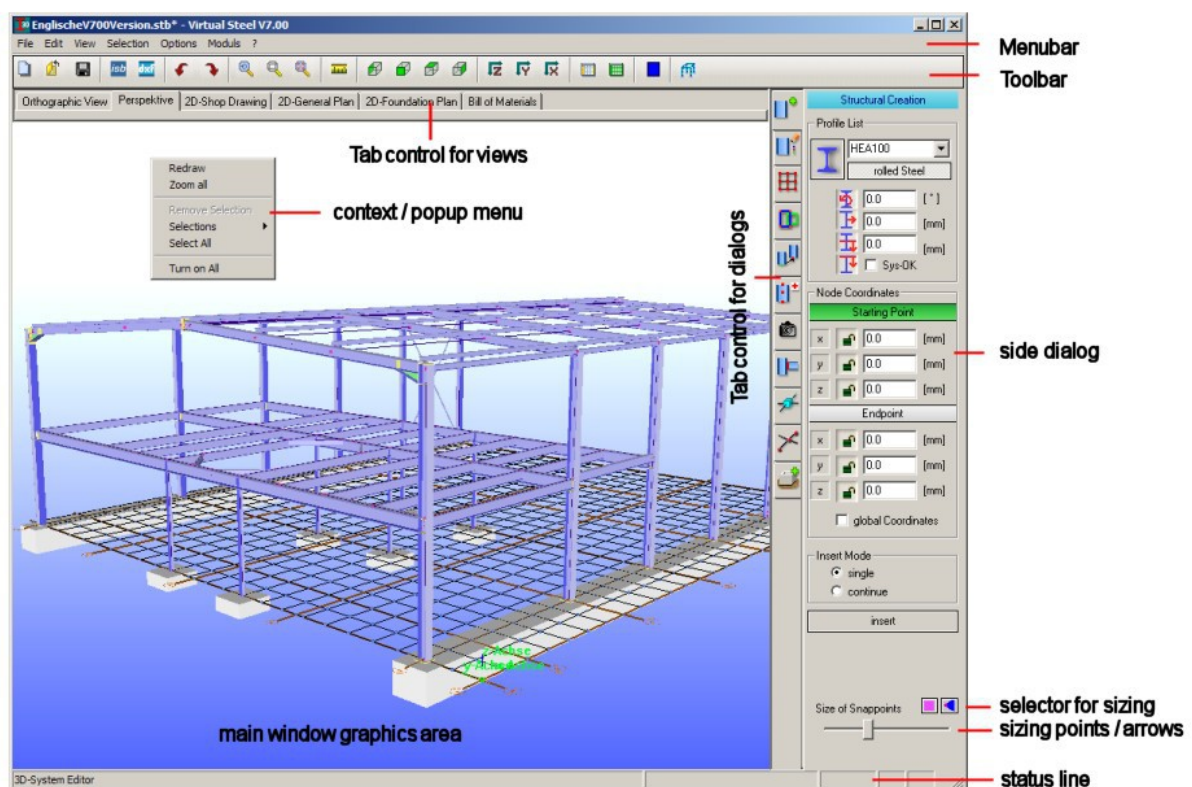
copy connections
copy structural elements
with connections

The workflow is illustrated through five numbered screenshots of the EngiSoft/WinStress software interface:

- 1. Create the structure / model:** Shows the initial 3D wireframe model of a building frame.
- 2. Create the connections:** Shows the 'Connections' dialog box where connection types (e.g., 'Explosion of a beam') are defined for the model.
- 3. Edit the details:** Shows the 'LookUp - Additional Part' dialog box, used for defining specific details or materials for the structure.
- 4. Numbering and editing the BoM:** Shows the 'Bill of Materials' (BoM) table, which lists all structural elements and their properties. The table includes columns for Number, Qant, Section, length [mm], Remarks, Grade, 2Dh - 2Dl, weight [kg], and Area. The table is populated with data for various structural elements like HEA 300, HEA 200, and HEA 100.
- 5. Export/print 2D drawings and BoM:** Shows the final 2D drawings and the BoM table, indicating the completion of the workflow.

The User Interface

As mentioned in the workflow already, the main window is controlled by a tab control. Another tab control, which is arranged vertically, is switching between different dialogs on the right side. These dialogs are also changing automatically, if certain commands need informations for their actions. With a right mouse-click in the main window, a context menu is shown, which is containing items of the object, which was hit by the mouse cursor. If no object got hit, a general popup menu is displayed, containing options of the current view.

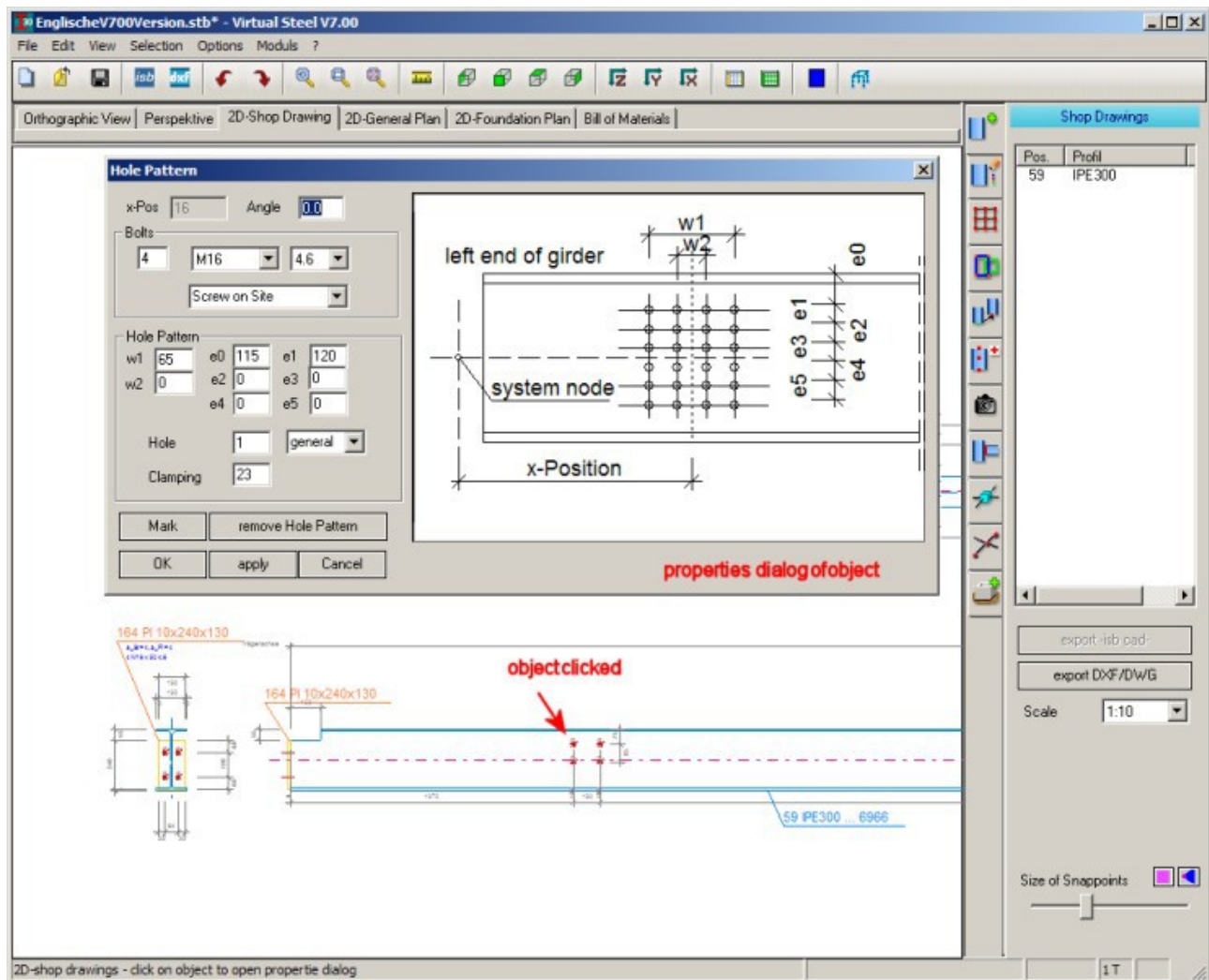


The typical menubar and a toolbar are located at the top of the window. Most commands of the toolbar are also available in one of the pulldown menus of the menubar.

As points in 3D may be next or far away from the user, only the pink and light blue points can be selected by a mouse-click in the graphics window. The size of these points can be changed with a scrollbar in the right side dialog at the bottom.

The mouse-clicks are acting differently in 3D and in 2D. In 3D an object is going to be selected with a simple left mouse-click, in 2D a dialog will show up with the properties of the object, which was hit by the mouse cursor. In 3D multiple objects can be selected by pressing the Ctrl-key simultaneously or dragging a window across the screen, in 2D only one object can be active by showing up the properties dialog. A right click in 3D opens up a context menu, in 2D there is only a

popup-menu for welding symbols and zoom functions.



Zooming in and out is similar in 2D and 3D, when the mouse wheel is used. In 3D there is no function for zoom window, just for zoom all. For a complete list of mouse actions combined with a pressed shift or ctrl key can be printed out from appendix A in the document.

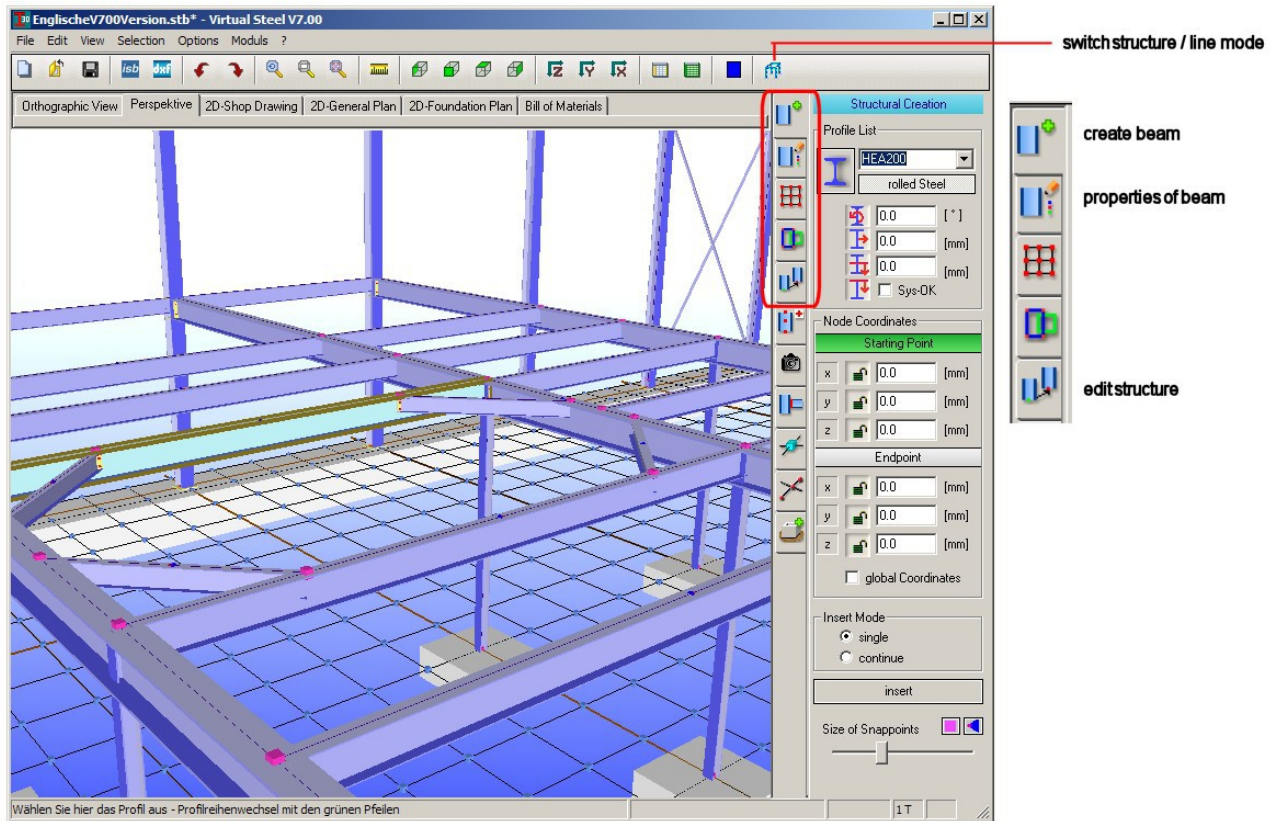
Creating the Structure

The structure consists of beams and basement objects. The basements are just for grounding the steel objects with anchor plates and anchor bolts. They are not of the construction process.

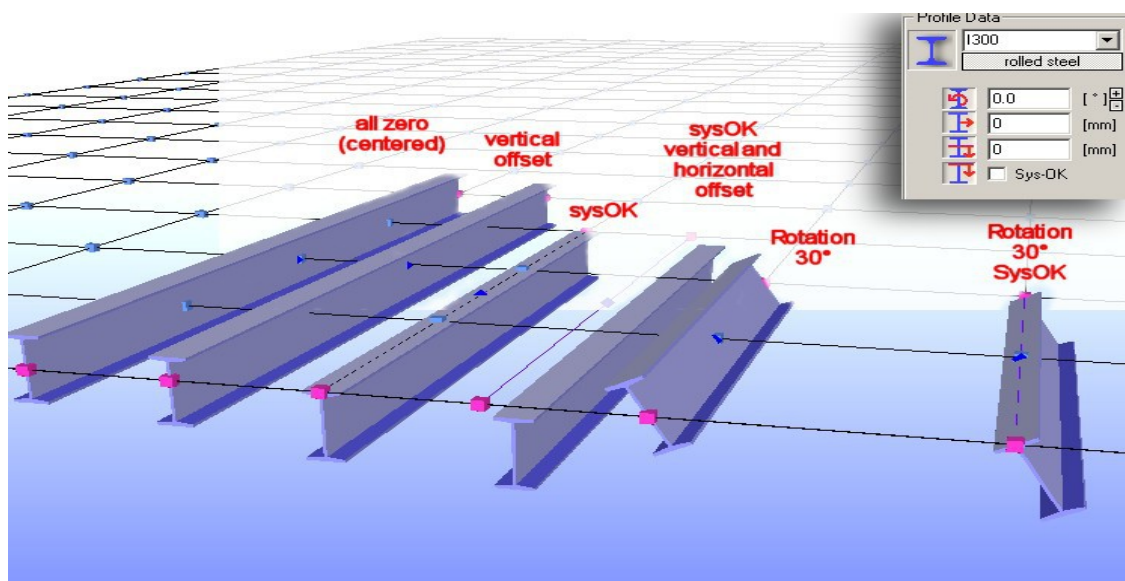
Beams

Beams have a starting and an endpoint each. Between these two points a definition line is visible with a blue-to-red arrow, showing the direction of the beam. The beam can be moved away from this line and rotated around this line. These local displacements can be defined when the beam is inserted or changed later in the properties dialog.

The dialogs, which are used for creating and editing the elements of the structure are shown in this graphic. The properties dialog, which is the second tab from above, will also open, when the menu item *properties* in the context menu is clicked.



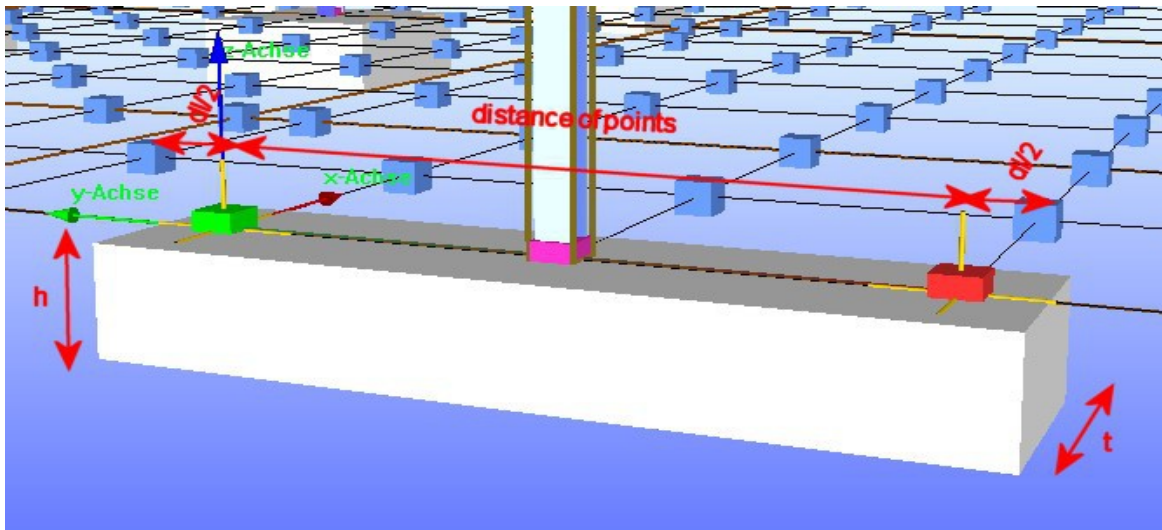
In the sample below there are shown different offsets and combinations of offset and rotation.



Basement Objects

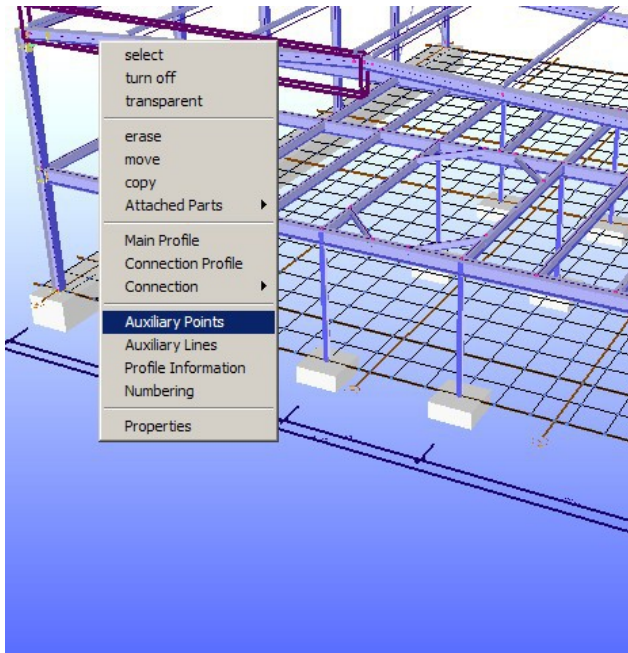
Foundations are just rectangular blocks and constructed similar to the beams. Once inserted, they can not get changed, only deleted. The dialog is likewise to the beams construction. On top the size of the foundation can be entered and a local offset horizontal and vertical.

In contrast to profiles a single point is sufficient to create a basement object. Then dl is the length, h the height and t the thickness of the object. If two points are chosen, dl is divided and added to the distance of the two points.



Help Points and Help Lines

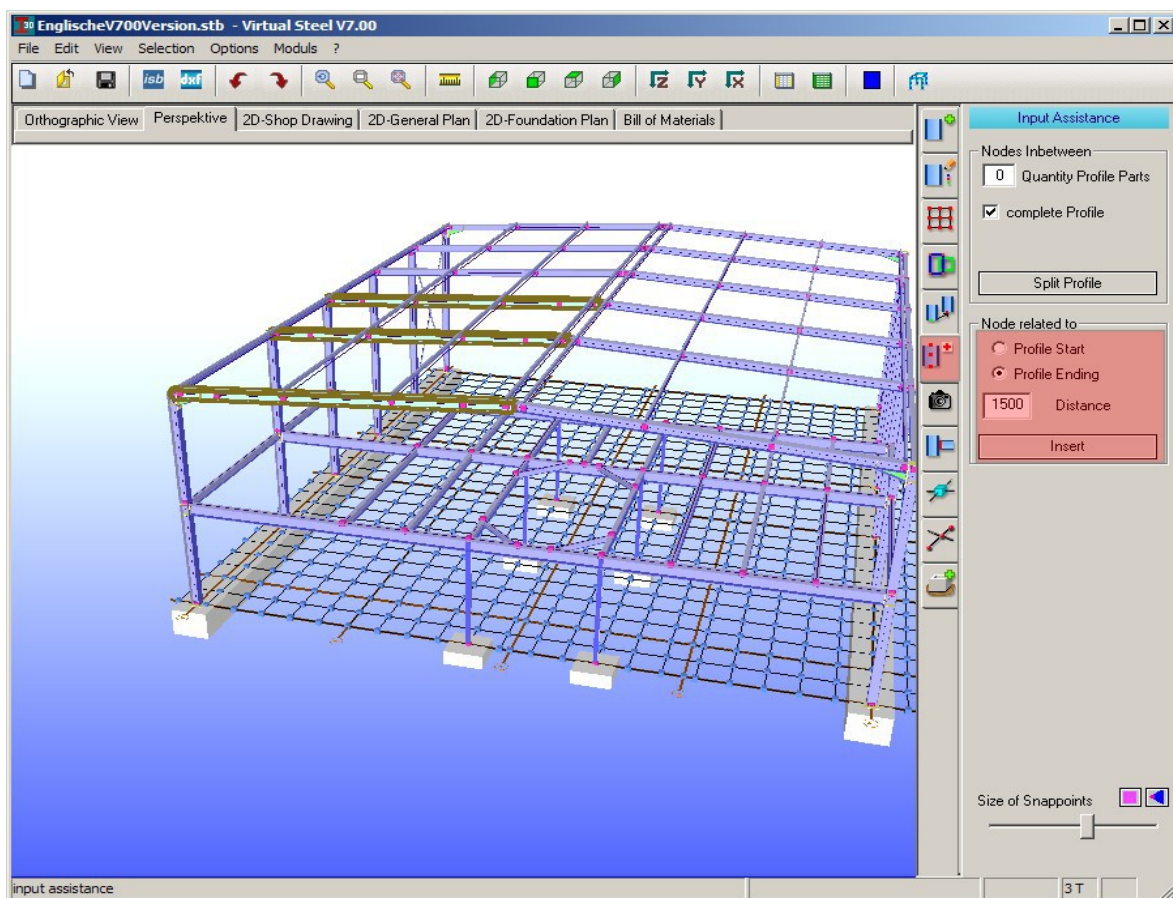
As mentioned at the beginning, the lines of the beams have to intersect, to connect the profiles. For this purpose a few options are available, to create help points. Help points also have a pink color like the endpoints of the beams. They stay in place until a *redraw*-command is triggered.



The most used command is, that a point in a certain distance from one of the endpoints is needed. To create such a point, open the context menu of the beam, on which the point shall be placed, click on Auxiliary Points and the side dialog will change.

This method is working for a single beam, but the points can also get created on multiple beams in one step. If the selection changes, they are created on all selected profiles in one step.

Moreover it is possible, to select a few profiles first, then click on the tab with the symbol for auxiliary points and then insert after applying the value for distance (1500 mm from the endpoint in the example below).

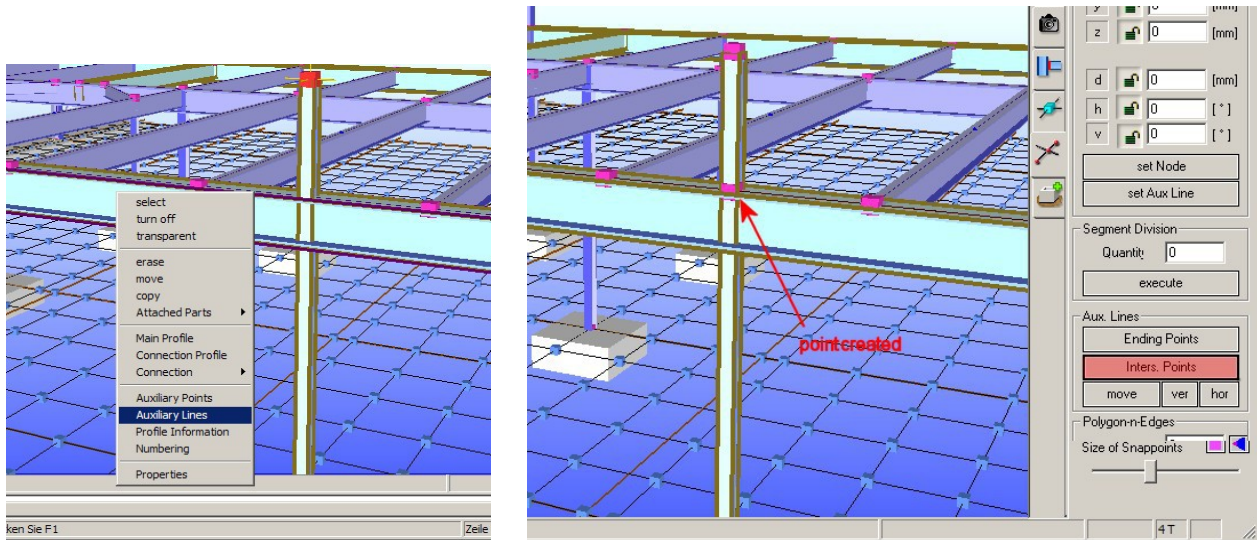


Intersection Points

An easy way to find the intersection point of two beams is to select both beams, open the context menu of one of them and select Auxiliary Lines. The right side dialog will change and offer a button named *Inters Point*. A click on this button will insert a point at the intersection. If the profiles have no intersection, no point is created.

Help Lines

	Coordinates Point 1	<p>The side dialog is opened by click on the tab with the symbol for auxiliary lines. Nodes, which are hit by a left mouse-click will show up their coordinates in the fields which are activated, shown by the green highlighted label. To activate the area, click on the gray label.</p> <p>With help lines geometrical data can be entered with distance and angles. The buttons beneath the target information area create auxiliary points or auxiliary lines. The lines have not endpoints, but can get them later by a click on the button <i>Ending Points</i>.</p>
	Coordinates Point 2	
	d - distance h - horizontal angle v - vertical angle	
	divide in parts of equal length	
	create points for lines and move them	
	create polygon with n edges	



Editing the Structure

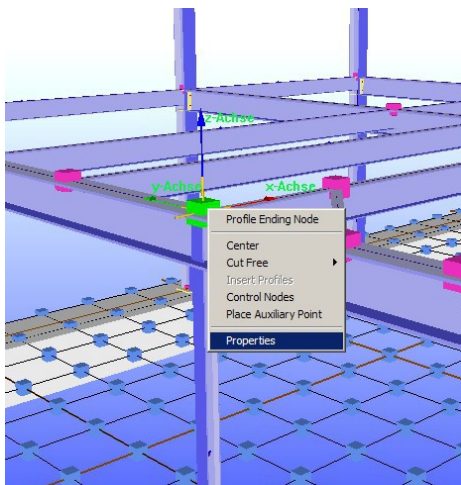
Structural Editor

Commonly used commands in CAD Systems are Delete, Copy, Move and Rotate. These commands are available in Virtual-Steel in the right side dialog. The command or action is selected at the top of the dialog and is then changing some elements and the behaviour of the buttons. There is another command named Stretch, which is needed for actions to extend a beam or multiple and keep all connections at their place. The new geometric position of a Copy or Move-action can be entered relative, so just as an offset of the origin elements has to get applied, or absolute. Absolute is the common way, if two points should define the offset. Multiple copies, each spaced by the distance between the points or the offset can get created in one step. Rotations are only done relative to the ground floor, the x-y-plane. The rotation point represents a vertical rotation axis.

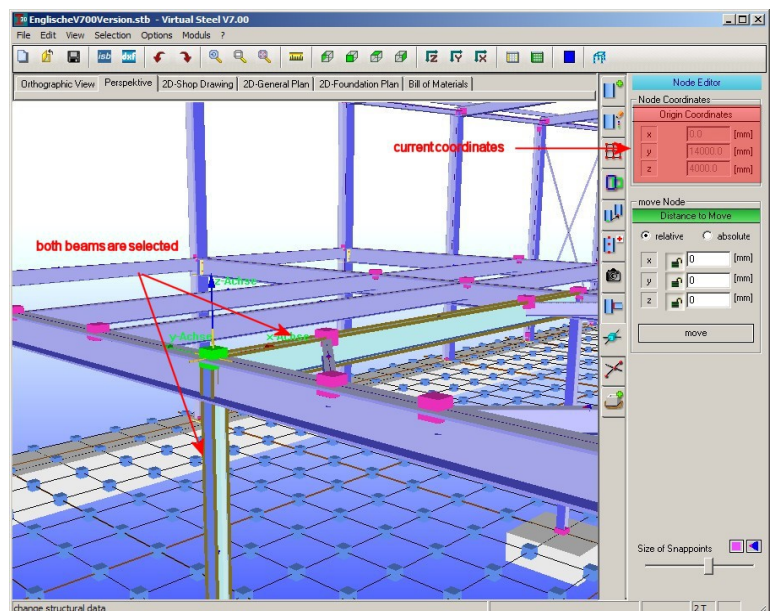
The Stretch function needs a bit more attention, as the type of selection is defining the area, which will get extended. The selection must be done by dragging a rectangular window from right to left across the parts, which are to be moved. All parts, which are complete inside the rect, are getting moved and at the border the extension will take place.

Adjustment of Points

Changing the coordinates of points will affect profiles, which have these points as ending points. Only one point at a time can be changed. To select a point, open the context menu with a right click on this point and click on properties.

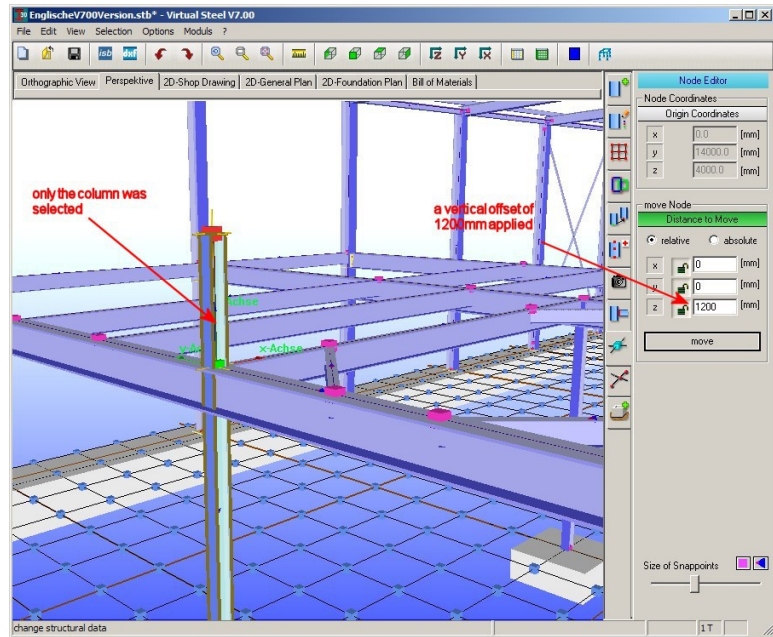


When clicking on properties, the right side dialog will display the current coordinates of this point and below that gives the user a chance, to enter an offset.



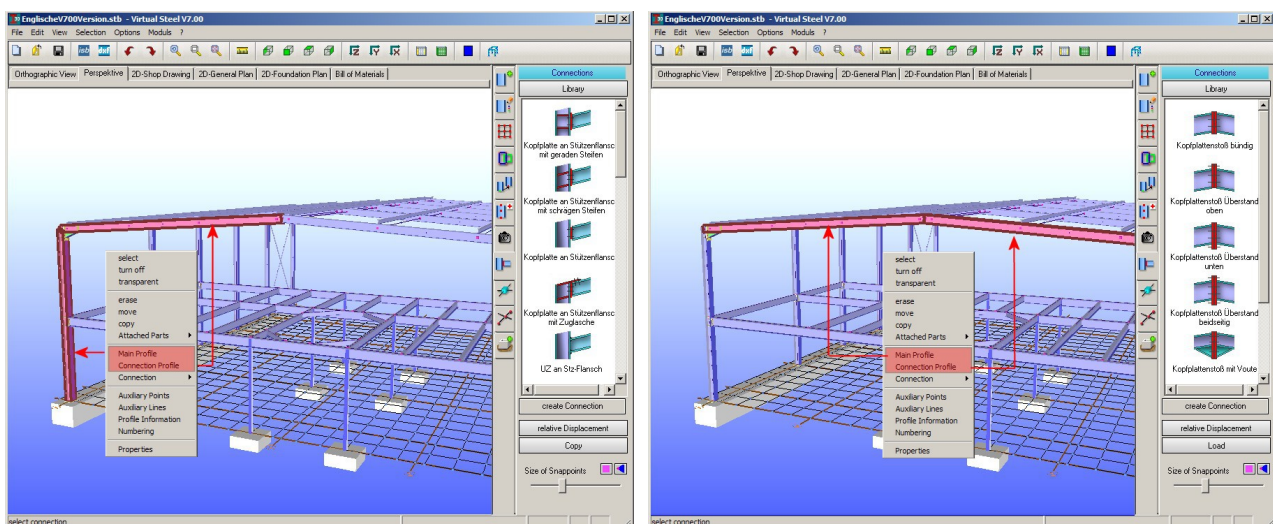
Relative means, that the new position will be the current coordinates with the offset added defined in x, y and z. Using the option *absolute* will move the point directly to the coordinates displayed, no matter whether entered manually or with a mouse-click.

All beams, which start or end at the point which was hit by opening the context menu, are selected automatically. If the displacement of the point should not affect all beams, those, which should stay in place can be deselected.



Creating Connections

To create a connection, two beams must be selected. To show that it is a different selection, those beams are getting colored in red. To select the beams there are two menu items in the context menu of the beams: *Main Profile* and *Connection Profile*. The main profile is the profile to which the connection profile is going to be connected. Sometimes both profiles have the same status. Then it is up to the user to define one of them as main profile.



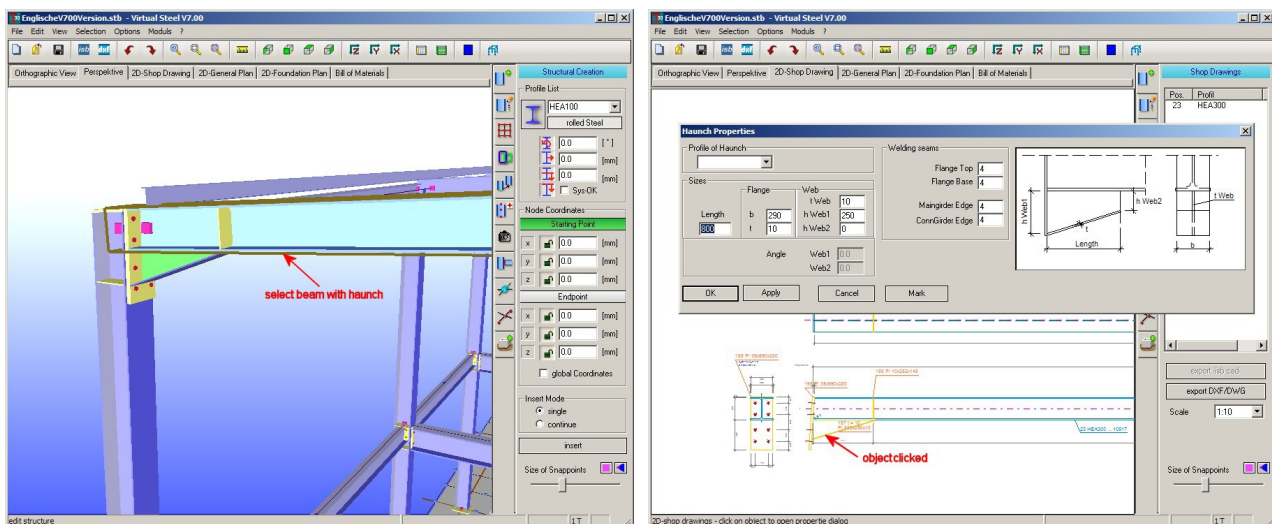
When the two profiles are highlighted in red, the right side dialog will change to the connection dialog, containing a library with all suitable connections. If the list of connections is empty, the profiles can not get connected. In most cases there is no intersecting point. The structure should

be checked by clicking on the icon in the toolbar, which is showing the definition lines only.

Editing the Details

Object Properties

Every object is described by a lot of different attributes. Mentioned above are some important attributes, which describe the dimensions and the displacement from the definition line. But beams can also get sawed at their ends in a horizontal or vertical angle. They can get equipped with drill holes, stiffeners, haunches etc. Other objects like plates, haunches, cleats, bolts have their individual dimensions and other properties. All these values can get accessed by clicking on the object of interest in the *2D-Shop Drawing*. The necessary steps are: select the beam to change or if it is the beam, where the object is attached to in the 3D-View, switch to the 2D-Shop Drawing and click on the object, which should get changed.



The reason, that these properties are made available in the 2D shop drawing, is the advantage, that the dimensions are shown there. Changing any value there will update the drawing and will then get visible in 3D too, when the user changes into 3D-mode.

Additional Parts

Inserting Additional Parts

If a profile's properties dialog is opened in the 2D Shop Drawing, there is a selection box, which offers additional parts to be added to the profile. These parts are stiffeners, endplates, anchor plates, bolts, bracings – all depending from the kind of cross section. The parts are added manually. For bolts, that are used for connections, the better way is always to create them with a

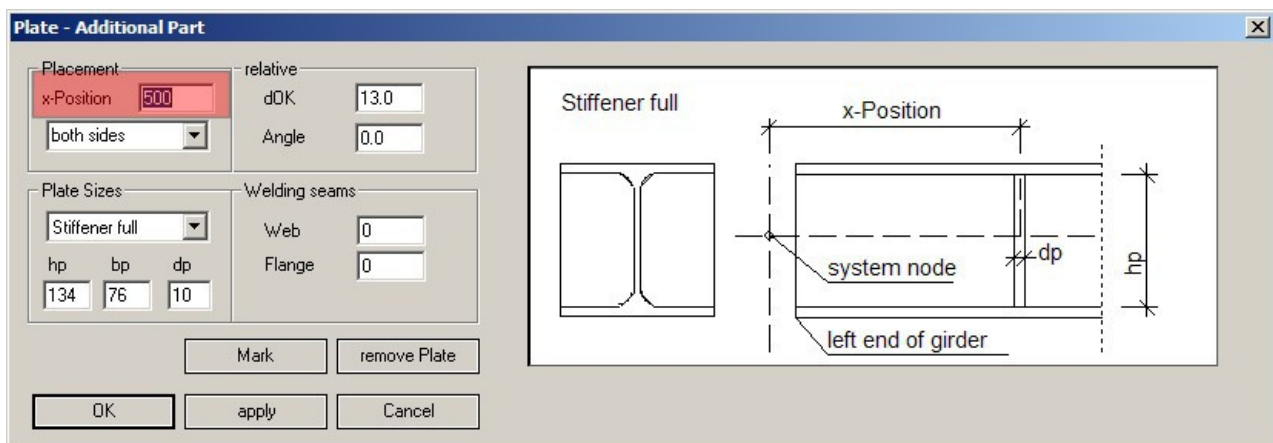
connection, as the drill holes will be in both profiles. By adding a hole pattern manually, only the profile is affected, on which this action was applied to.



First select the item, then click on insert to add it to the profile

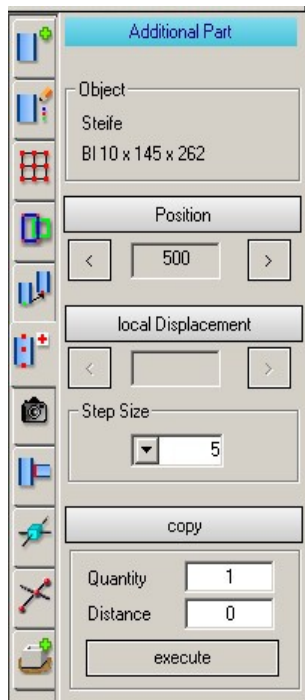
Each additional part or hole pattern has a value for its placement in the profile. The value is called x-Position and has the zero point at the beginning of the raw profile. Raw profile means, that the truncation (or an extension) is disregarded. Just keep in mind, that this position is actually a global position concerning the whole model and not just the profile it is attached to. Sometimes the edit field of the x-Position is grayed. This is the case, if the part belongs to a connection or if it has a counterpart in another profile.

A graphic on the right side of the dialog is intended to clarify the values of the dialog.



Moving Additional Parts

Moving the additional parts can be done by changing the values in the dialogs of the shop drawing, but it can also be done in the model.

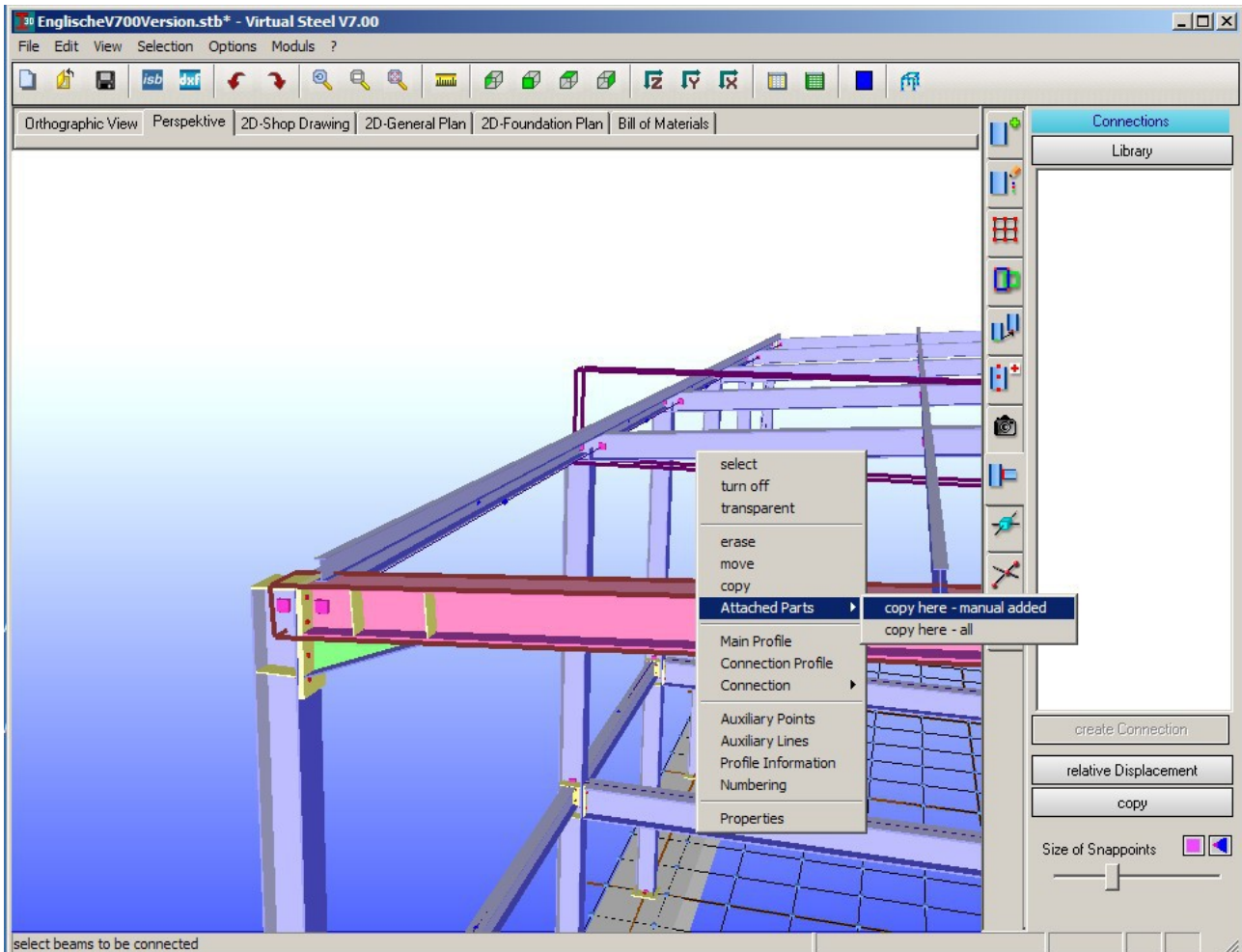


Opening the context menu of the object and clicking on *Properties* will show up a dialog with informations and some buttons on the right side. Using the arrow keys will move the object in which the increment value is taken by the selection box below – the *Step Size*. Sometimes the Position is offered, sometimes the local Displacement. It depends on the current object.

Copy Additional Parts

There are two options to copy additional parts and hole patterns. One is shown in the graphic above. In this case, the object will be copied to the same profile in a specified *Distance*.

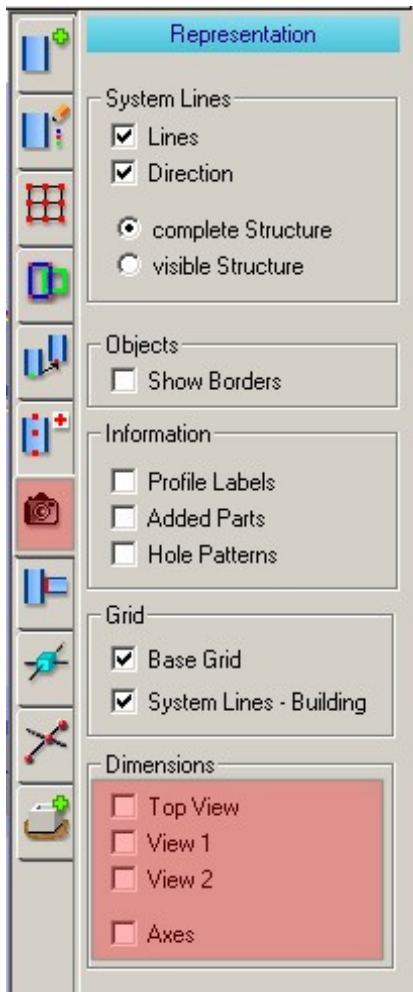
The other one is significant for copying added parts from one profile to another. In this case the source must be selected first. It is selected as a *Main Profile* – context menu of a beam, clicking on *Main Profile*. Then bring up the context menu of the target profile by rightclicking the mouse and select the menu item *Attached Parts – copy here – manually added*, or the item beneath.



The graphic is showing the selected source profile highlighted in red and the target profile in the background with its context menu.

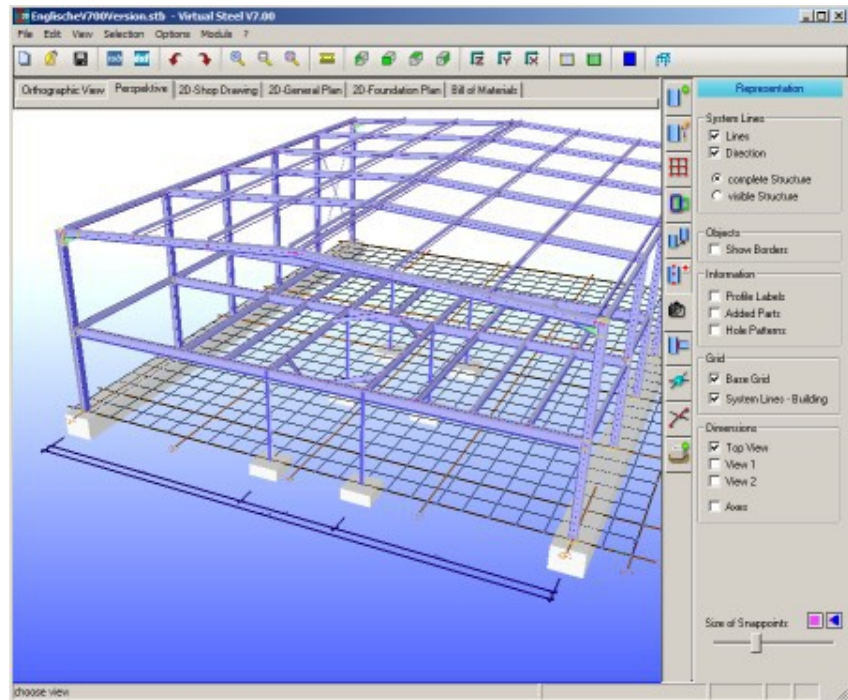
2D-Drawings

In the chapters before, 2D-Shop Drawings were discussed already. To show an object in the shop drawing, it must be selected. The other 2D drawings do not require a selection. If nothing is selected, all elements are shown in the general plan and all foundations in the 2D Foundation Plan. If, however, objects are selected, only these selected objects are shown in the 2D General Plan or the Foundation Plan. The Foundation plan has the peculiarity, that basements objects are to be selected and that only those columns are included, which have joints with the basements.



To show dimensions in the general plan, open the side dialog 3D viewing properties and select the views for the dimensions.

The dimensions are only getting attached to the current selection. If nothing is selected, all relevant points are dimensioned.



Bill of Materials

The Bill of Materials is always up to date, as it is generated, when the table is getting displayed. So changes are reflected immediately. When the view is switching to the BoM table, the right side menu will show the BoM dialog. Here the user can manually start an analysis, which will try to identify all identical parts, which then are supplied with equal numbers. The objects with equal numbers will get counted and this is shown in the columns of the quantity. As the renumbering will generate gaps, there is a button, which helps the user to eliminate these gaps.

Number	Qunt	Section	Length	Remarks	Grade	DIN - EN	Weight [kg]	Area
1	8	HEA300	10688		S235JR		7984.632	147.0
...	(7)							
9	2	HEB160	7000		S235JR		596.757	12.8
10	8	HEB160	5000		S235JR		1705.020	36.7
...	(8)							
19	32	IPE140	5000		S235JR		2059.940	88.1
20	8	IPE140	7000		S235JR		720.944	30.8
...	(38)							
59	7	IPE300	7000		S235JR		2069.417	56.8
60	8	IPE300	5000		S235JR		1689.320	46.4
...	(12)							
71	4	IPE200	2121		S235JR		189.808	6.5
...	(3)							
77	4	HEA300	10517		S235JR		3731.642	72.3
...	(2)							
80	6	HEB140	4000		S235JR		810.120	19.3
...	(4)							
85	2	HEA300	7975		S235JR		1414.845	27.4
...	(1)							
87	1	HEA300	8000		S235JR		709.640	13.7
88	2	L70*50*6	8834		S235JR		95.699	4.2
89	2	L70*50*6	8834		S235JR		95.699	4.2
90	1	HEA300	8110		S235JR		719.398	13.9
...	(1)							
92	1	HEA300	8110		S235JR		719.398	13.9
...	(1)							
94	1	HEA300	7975		S235JR		707.422	13.7
95	1	HEA300	8110		S235JR		719.398	13.9
96	3	HEB400	20982		S235JR		9783.697	121.4
...	(1)							

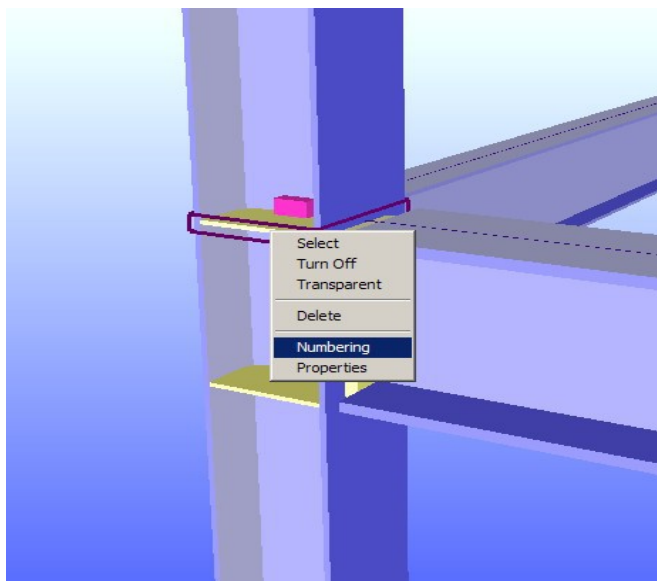
Highlighted gaps after *identical parts* were gathered and *edit gaps* fired.

To close this gaps, bring up the context menu by clicking on the gray columns with right mouse-click. If more than on row is selected, all included gaps are eliminated by clicking on *move up table rows*.

Number	Qunt	Section	Length
80	6	HEB140	
...	(7)		
...	(7)		

Number	Qunt	Section	Length
1	8	HEA300	
...	(7)		
8	2	HEB160	
...	(7)		
19	32	IPE140	
20	8	IPE140	
...	(38)		
59	7	IPE300	
60	8	IPE300	
...	(12)		

The numbers can be changed by just clicking into the cell with the number and entering the new number. All elements with this number are changed together. If just one element should receive a different number, this must be done by clicking on the element in the 3D view with the right mouse button to bring up the context menu and choose numbering in this menu.



BOM Number Edit

Caution!!
Entered numbers are not getting checked for individuality.

BOM Number: 162

OK Cancel

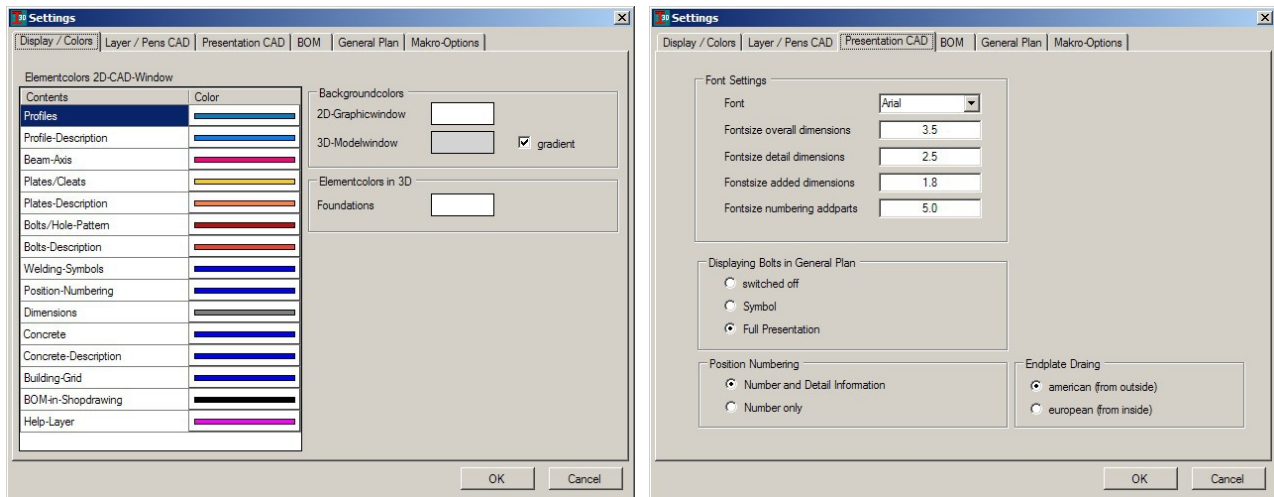
The current state of numbering can be set to constant. This is important, if the numbering may not change, due to the fact, that a section of the building is already going to be fabricated. If new parts are created, these will always get new numbers. Even an analysis of identical parts will not add

parts of the new section to those, which are identical but in the constant section.

Extras

Configuration and Settings

Colors, fonts and sizes can be chosen in a special settings dialog. This is done once to configure the Software for the user's purpose.



Project Manager

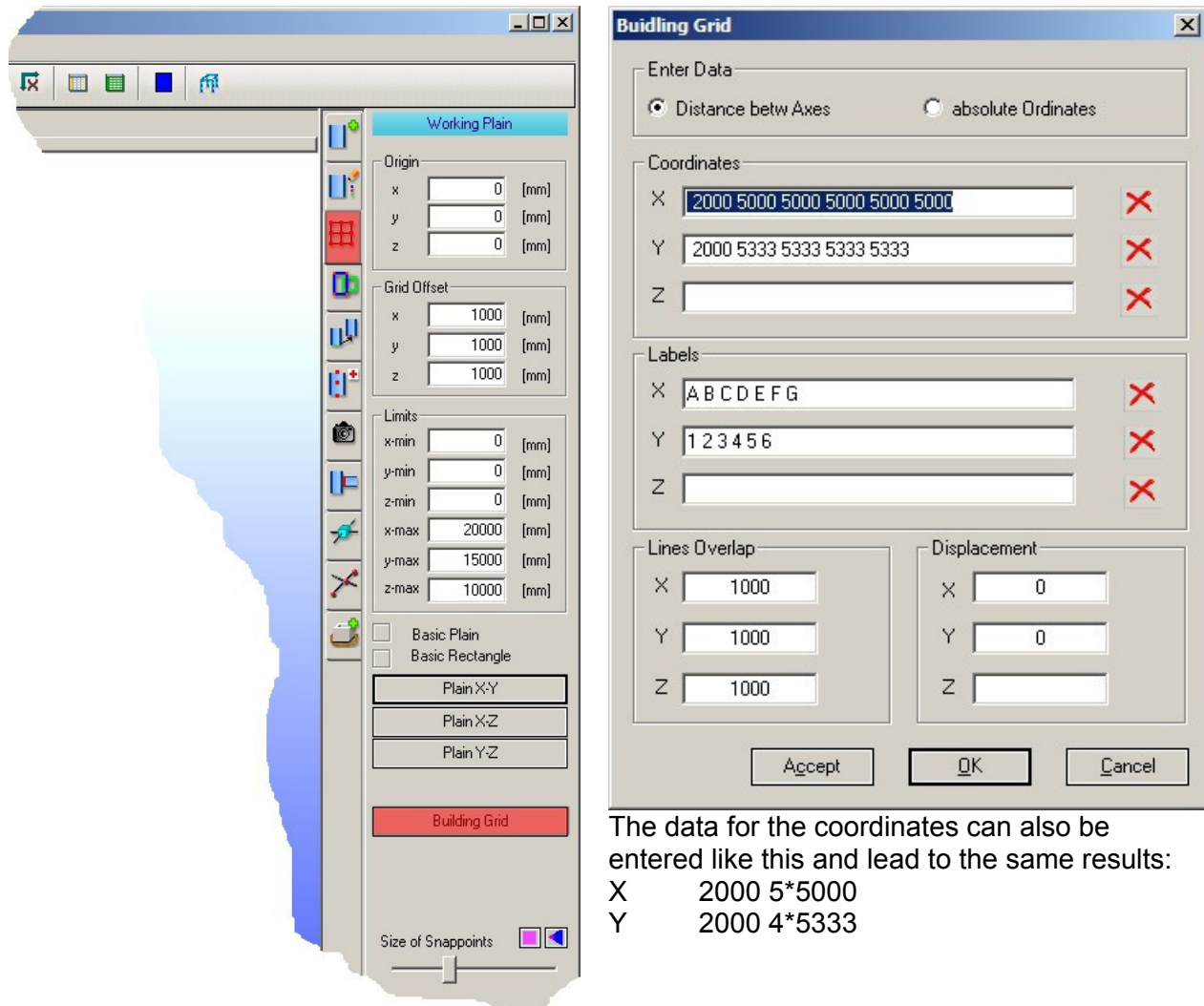
The project manager keeps track of projects, which were recently edited and saved. Each project is saved with a image for easier identification. Just the names of the projects are displayed, nit the folders where they are stored in.

Project Informations

In the file menu a menu item allows to set project informations. Thes informations are used when printing the BoM or drawings

Building Grid

The building grid is defined in a dialog which is opened by a button in the right side dialog. The side dialog allows different settings for the grid, the size in x and y, the origin and the limits.



Import and Export

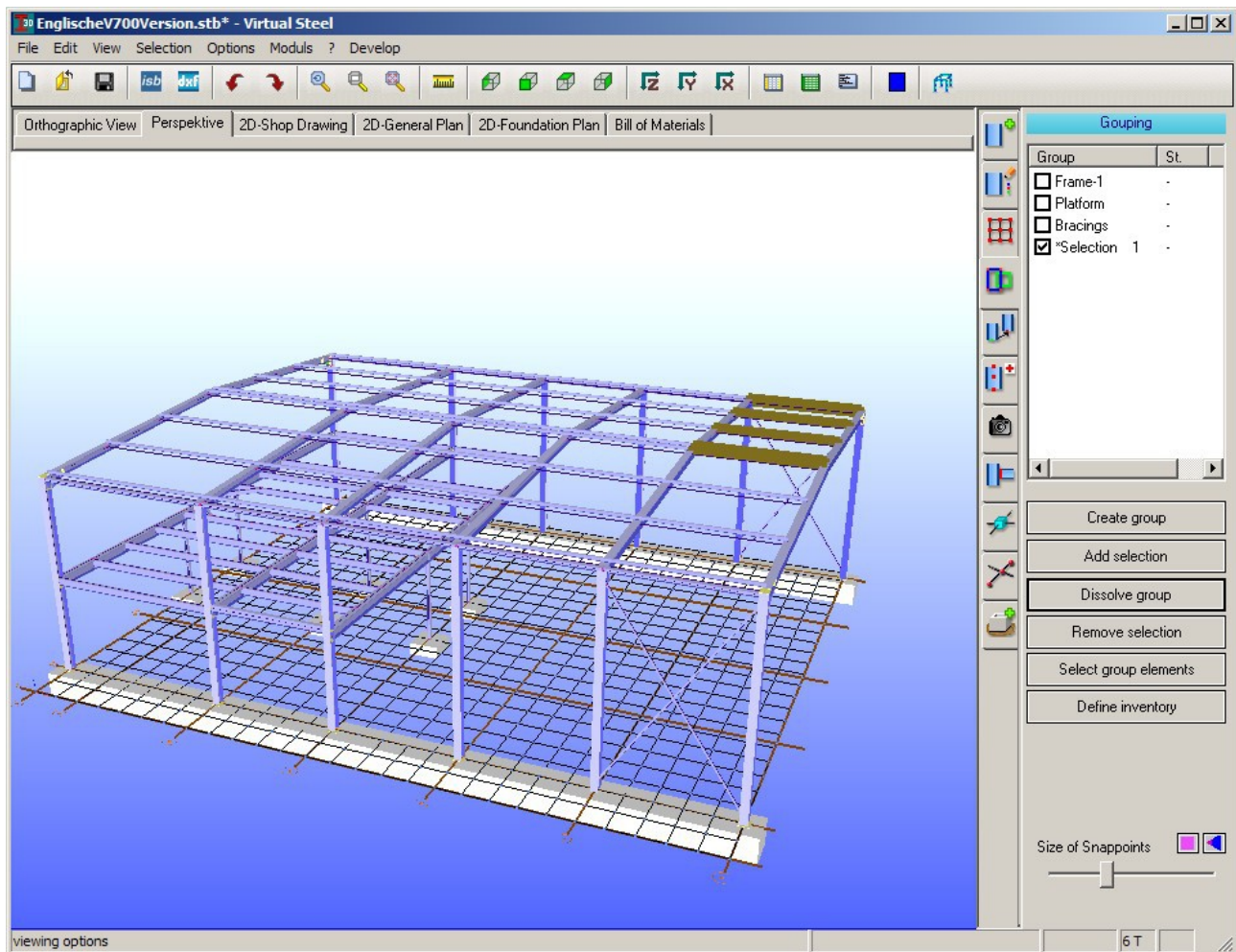
Several file formats can be read and written to exchange graphics or project data. Most of them are gathered in the *File* menu, but 3D-DXF and the Excel-export are available through buttons in the right side dialogs for 2D-General plan and Bill of Materials.

Groups

To make the model more clearly and to arrange the objects, you can set up groups. Every beam can get stored in one or more groups. A column can be in the group “Columns” and also in the group “Frames”. The Group Inventory represents a particular group and can be used as part of the building, which is already existent and should be modified or extended. The objects in this group are not included in the bill of materials.

Selections can be saved and will then get listed in the dialog. In contrast to normal groups the ticking of the box causes that the elements are displayed as a selection, while ticking the box of a group will only leave the elements in the group visible and switch off the others.

Groups and selections can be named by clicking on the label and holding down the mouse button for a short period of time. Then a name can be entered. A selection can be changed into a group, if the name is changed.



Appendix

Mouse Hotkeys Overview

The combinations of mouse buttons with **[ctrl]** and **[shift]** offer shortcuts for a quick handling of view and selection actions. After a short time working with Virtual Steel it is possible to act very efficiently with these button-key combinations. Here is a list of these combinations.

Mousebuttons	Keys on keyboard		
		Shift	Ctrl
Left Mousebutton			
Single click	Catch single object, identify snap point	Select <i>main profile</i> , selektieren (1. click), select <i>connection profile</i> (2. click)	Add to selection if not in selection or deselect, if already in selection
Drag window from left to right	All objects, which are entirely inside the rectangle are getting selected or added to an existing selection		
Drag window from right to left	All objects, which are inside or touched by the rectangle are getting selected if not in the current selection, otherwise they are deselected		All objects, which are inside or touched by the rectangle are getting added to the current selection
Middle Mousebutton			
Single click		Remove all selections	
Pressed and Moving	Move model		Left/right – turn model up/down – tilt model
Right Mousebutton			
Simple click	Open Popup-Menu – depending on the model which was hit (context sensitive)		Center view if clicked on a point or profile defining also the rotation center point
Pressed and Moving	Turn model		
Mousewheel			
	Zoom in/out		Size of snap point/arrows enlarging / diminishing

Middle mousebutton is meaning a pressed mousewheel.