



# ADVISIM<sup>3D</sup>

a brand of  ADVITEC

## User manual

Version 2.0

2/20/2024

© 2015 - 2024 ADVITEC Informatik GmbH

# Table of contents

<b>Part I Welcome</b>	<b>6</b>
<b>Part II First steps</b>	<b>7</b>
<b>Part III Contact</b>	<b>11</b>
<b>Part IV User documentation</b>	<b>12</b>
<b>1 Program structure</b>	<b>12</b>
<b>2 Quick Start Guide</b>	<b>12</b>
<b>3 Release Notes</b>	<b>13</b>
<b>4 Projects</b>	<b>25</b>
<b>4.1 Create new project</b>	<b>25</b>
<b>4.2 Save project</b>	<b>25</b>
4.2.1 Save simulation	25
<b>4.3 Open project</b>	<b>25</b>
4.3.1 Open simulation	27
<b>5 View control</b>	<b>27</b>
<b>5.1 Mouse interaction</b>	<b>28</b>
<b>5.2 Open context menu</b>	<b>30</b>
<b>5.3 Keyboard interaction</b>	<b>31</b>
<b>5.4 3D coordinate system</b>	<b>32</b>
<b>5.5 Structure tree</b>	<b>33</b>
<b>5.6 Simulation views</b>	<b>35</b>
<b>5.7 Up-direction of the model</b>	<b>35</b>
<b>5.8 Undo/Redo actions</b>	<b>36</b>
<b>6 Properties area</b>	<b>37</b>
<b>6.1 Properties area for nozzles</b>	<b>37</b>
6.1.1 Change names	39
6.1.2 Changing the type	39
6.1.3 Change position	40
6.1.4 Change direction	40
6.1.5 Angle to the normal of local reference planes	41
6.1.6 Change working pressure	41
6.1.7 Change up-vector	41
6.1.8 Change distance	42
6.1.9 General properties	42
<b>6.2 Clipping plane properties area</b>	<b>43</b>
6.2.1 Change names	44
6.2.2 Change position	44
6.2.3 Change normal	44
6.2.4 Angle to local reference planes	45

6.2.5	General clipping plane properties .....	45
<b>6.3</b>	<b>Custom view properties area .....</b>	<b>45</b>
<b>6.4</b>	<b>Constraints properties area .....</b>	<b>46</b>
6.4.1	Show name .....	47
6.4.2	Change orientation.....	48
6.4.3	Change distance.....	48
6.4.4	General properties.....	49
<b>6.5</b>	<b>Simulation properties area .....</b>	<b>49</b>
<b>7</b>	<b>Toolbar.....</b>	<b>50</b>
7.1	Take a screenshot .....	50
<b>8</b>	<b>3D toolbar.....</b>	<b>50</b>
8.1	Set visibility of all scene elements .....	51
8.2	Switch view mode .....	51
8.3	Standard views .....	52
8.4	Coordinate system planes .....	53
<b>9</b>	<b>Color scale area .....</b>	<b>54</b>
9.1	Color pipette .....	55
<b>10</b>	<b>Model part interaction.....</b>	<b>56</b>
10.1	Model import .....	56
10.2	Convert model part to nozzle .....	57
<b>11</b>	<b>Nozzle interaction.....</b>	<b>58</b>
11.1	Nozzle category .....	58
11.1.1	Target stream cleaner.....	58
11.1.2	Spray balls .....	59
<b>11.2</b>	<b>Insert nozzle .....</b>	<b>61</b>
11.2.1	Nozzle selection dialog.....	62
<b>11.3</b>	<b>Move nozzle .....</b>	<b>63</b>
11.3.1	Move nozzle along its spray direction.....	64
11.3.2	Move nozzle with constant target point.....	65
<b>11.4</b>	<b>Change nozzle direction .....</b>	<b>66</b>
11.4.1	Reset nozzle direction.....	66
11.4.2	Invert nozzle spray direction.....	67
11.4.3	Turn nozzle around its spray direction.....	68
<b>11.5</b>	<b>Select nozzle .....</b>	<b>68</b>
<b>11.6</b>	<b>Show and hide nozzle .....</b>	<b>69</b>
<b>11.7</b>	<b>Change nozzle type .....</b>	<b>69</b>
<b>11.8</b>	<b>Delete nozzle .....</b>	<b>70</b>
<b>11.9</b>	<b>Export nozzle .....</b>	<b>71</b>
<b>11.10</b>	<b>Show working range .....</b>	<b>74</b>
<b>11.11</b>	<b>Customize and create spray balls .....</b>	<b>77</b>
11.11.1	First steps .....	77
11.11.2	Input masks .....	79
	Nozzle type template.....	80
	General nozzle parameters.....	80
	Spray ball parameters.....	83
11.11.3	Edit spray balls.....	85
11.11.4	Delete spray balls.....	87

11.11.5	Exclusivity of custom spray balls .....	87
<b>12</b>	<b>Clipping plane interaction.....</b>	<b>87</b>
12.1	Insert clipping plane .....	87
12.2	Delete clipping plane .....	89
12.3	Move clipping plane along its orientation .....	90
12.4	Show and hide clipping planes .....	92
12.5	Select clipping plane .....	93
12.6	Tilt clipping plane .....	94
12.7	Invert clipping plane normal .....	95
12.8	Mark clipping planes .....	96
<b>13</b>	<b>Constraint interaction.....</b>	<b>97</b>
13.1	Constraint types .....	97
13.1.1	Parallel to a model part plane.....	97
13.1.2	Along a model part axis .....	98
13.2	Insert constraint .....	99
13.2.1	Add along a model part axis .....	99
13.2.2	Add parallel to a model part plane.....	101
13.3	Error message for invalid constraint combination .....	101
13.4	Delete constraint .....	102
<b>14</b>	<b>Motion interaction.....</b>	<b>104</b>
14.1	Add motion .....	104
14.2	Linear motion .....	104
14.3	Rotating motion .....	106
14.4	Show motion path .....	108
14.5	Simulate motion .....	108
14.6	Delete motion .....	109
<b>15</b>	<b>Model movement.....</b>	<b>109</b>
15.1	Set movement .....	110
15.2	Show movement line .....	110
<b>16</b>	<b>Custom camera views.....</b>	<b>111</b>
16.1	Insert custom camera views .....	111
16.2	Change custom camera view name .....	112
16.3	Delete custom camera view .....	112
<b>17</b>	<b>Simulation.....</b>	<b>113</b>
17.1	Start and continue simulation .....	113
17.2	Stop simulation .....	114
17.3	Configure simulation .....	114
17.4	Result view .....	115
17.5	Exit simulation .....	116
17.6	Time navigation .....	116
17.6.1	Playback mode.....	117
17.6.2	Quick navigation.....	117
17.7	Simulation modes .....	117
17.7.1	Direct volume application including footprint.....	118
17.7.2	Ablated layer thickness .....	118
<b>18</b>	<b>Print report.....</b>	<b>119</b>



18.1	Page settings .....	121
18.2	General information .....	121
18.3	Nozzle information .....	121
18.4	Non-visible model parts .....	122
18.5	Simulation information .....	122
18.6	Choose 3D views for report .....	123
19	Settings.....	123
19.1	Logging .....	125
19.2	Print report .....	125
19.3	Simulation .....	125
20	License management.....	125
20.1	Activate license .....	126
20.2	Show license information .....	129
20.3	License service .....	131
<b>Part V System documentation</b>		<b>132</b>
1	System requirements.....	132
<b>Part VI FAQs</b>		<b>133</b>
1	Spray pattern not visible.....	133
2	Firewall request at startup.....	135
3	Version expired.....	135
4	Unexpected simulation results.....	136
5	Known issues.....	136
5.1	Transfer of simulation results .....	136
5.2	Incorrect display of clipping planes .....	137
5.3	Localization of nozzle generated from model part .....	137
5.4	Pressure change of nozzle has no effect on workflow area .....	137
<b>Part VII Index</b>		<b>138</b>

# I Welcome



# ADVISIM<sup>3D</sup>

... to the ADVISIM<sup>3D</sup> Community!

We are glad that you have decided to use ADVISIM<sup>3D</sup>.

ADVISIM<sup>3D</sup> is your virtualization solution for the realistic simulation of your complex spray cleaning systems for the identification of volume flow distribution and cleaning effect fast and reliable at your office PC.

This guide provides all the information you need to get started quickly with cleaning simulation using ADVISIM<sup>3D</sup>.

You want to proceed methodically? Our [first steps](#) will support you.

You already use ADVISIM<sup>3D</sup> and are looking for a single function? With the [keyword index](#) you will get to the point.

Please do not hesitate to contact us for further information.

Answers to licensing, usage as well as purchase of ADVISIM<sup>3D</sup> but also to technical questions we have provided you under the following links

<https://www.advisim3d.com/eula>

<https://www.advisim3d.com/gtc>

<https://www.advisim3d.com/support>

We share news about ADVITEC and ADVISIM<sup>3D</sup> with you at

<https://www.advisim3d.com/news>

<https://www.advisim3d.com/messen-termine>

We are looking forward to your successful projects and your experience reports.

Have fun with ADVISIM<sup>3D</sup>!

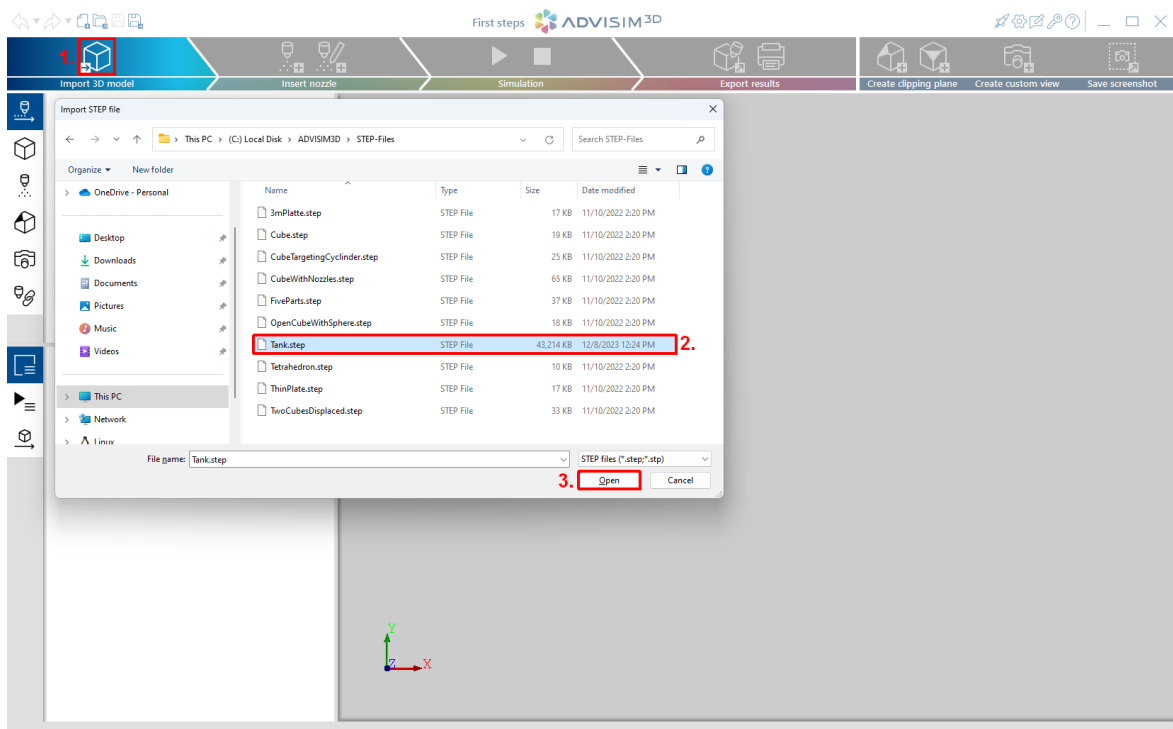
## II First steps

To give you a quick start with ADVISIM<sup>3D</sup>, the basic procedure is described here.

### 1. Import STEP file

After you have opened ADVISIM<sup>3D</sup>, the first thing you need is a [model in STEP file format](#).

To do so, select [Import 3D model](#) from the workflow bar or the Quick Start Guide and open the desired file.

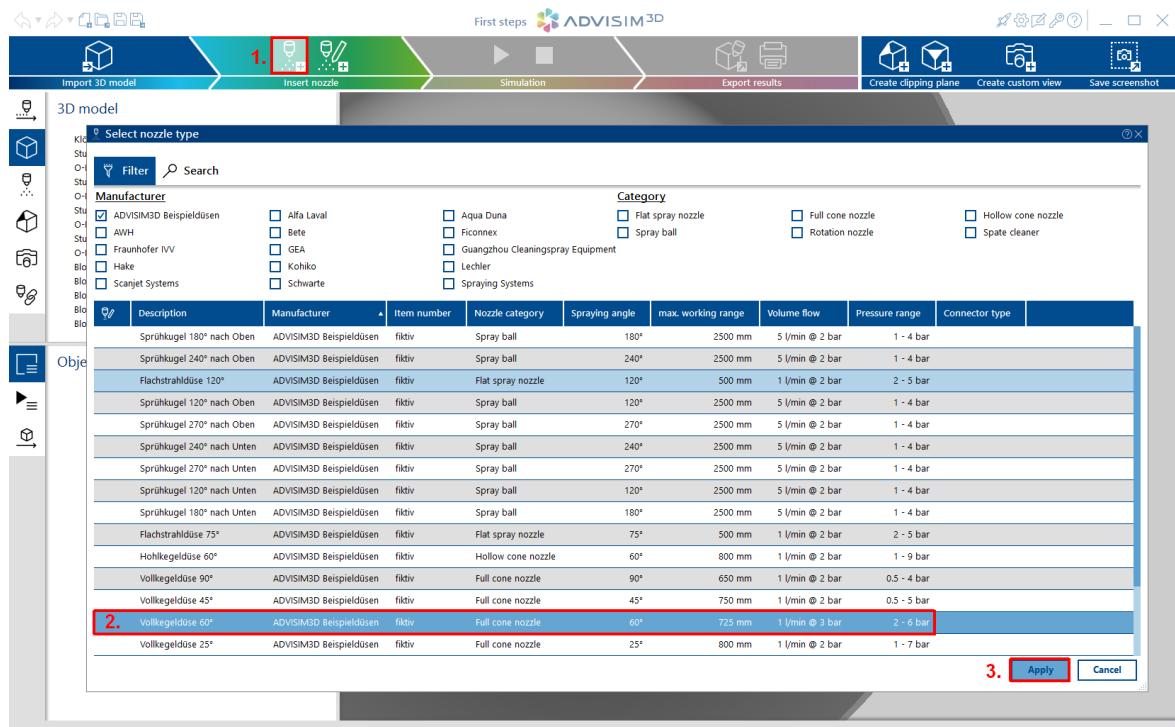


### 2. Select nozzle type

After you have successfully loaded a model, you can now [insert your first nozzle](#).

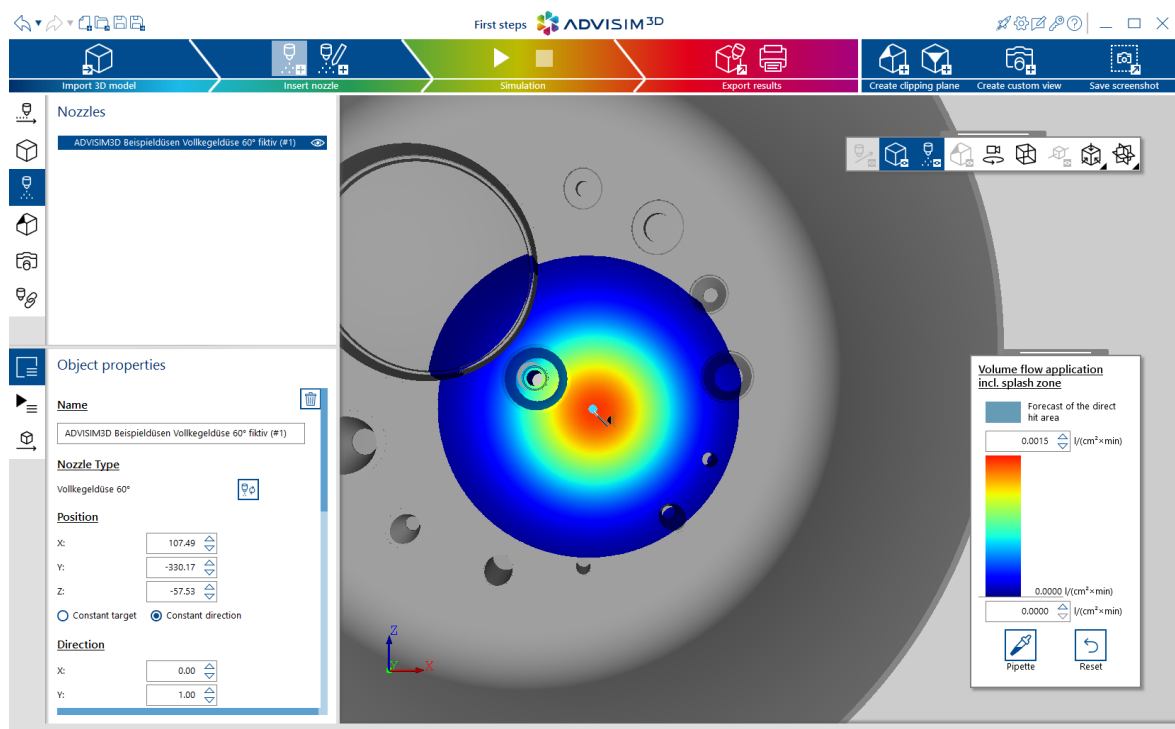
Select [Insert nozzle](#) from the title bar or the Quick Start Guide.

It is now possible to select nozzles in the [nozzle selection dialog](#) that opens. To narrow down the search results, you can filter the list of nozzle types by manufacturer and [nozzle category](#).



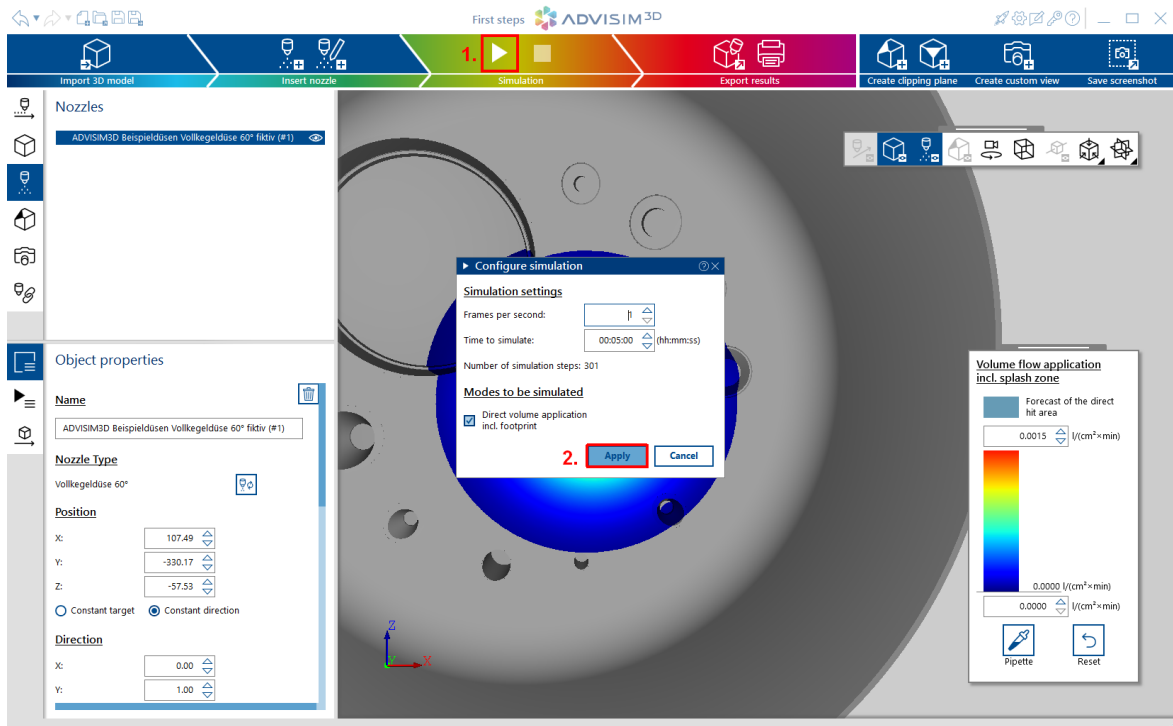
### 3. Insert nozzle

You are now in [Insert nozzle mode](#). Place the nozzle by pointing to the model in the 3D scene with the mouse and pressing the left mouse button.



### 4. Configure nozzle simulation

Then simulate the behavior of the nozzle over time. To do this, click on the "Start simulation" button in the title bar or the Quick Start Guide. The dialog for [configuring the simulation](#) opens. The input fields are filled with default values. Click on the "Apply" button.

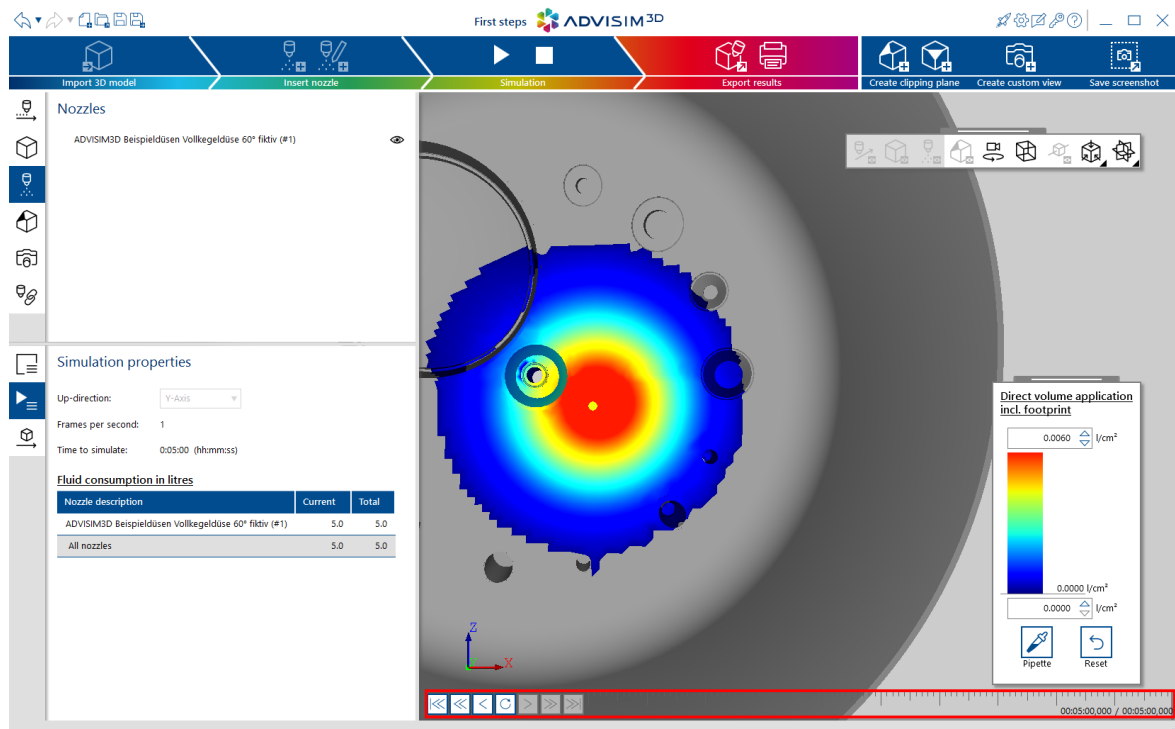


## 5. Simulation results

After the spray simulation of the nozzle has run for the specified period, the [results are displayed](#).

Now [move around in the 3D scene](#) and view the results from all sides.

Use the time navigation below the 3D scene to view the simulation results at different points in time. To do this, click on the desired position in the [navigation bar](#) or use the buttons to the left of it.



## Congratulations

You have performed your first simulation of a nozzle. There are many more options in ADVISIM<sup>3D</sup>.

As an example, you can

- select a different [nozzle type](#) in the [nozzle selection dialog](#) and simulate their different behavior
- create your [own spray balls](#)
- use [constraints](#) or [model part conversion](#) for simplified nozzle placement
- assign a [linear motion](#) to the model
- [save important views](#) and print them in a [print report or save them as PDF](#)
- use a [clipping plane](#) to simulate the internal cleaning of a model, for example.

### III    **Contact**

Please do not hesitate to contact us if you have any questions or suggestions.

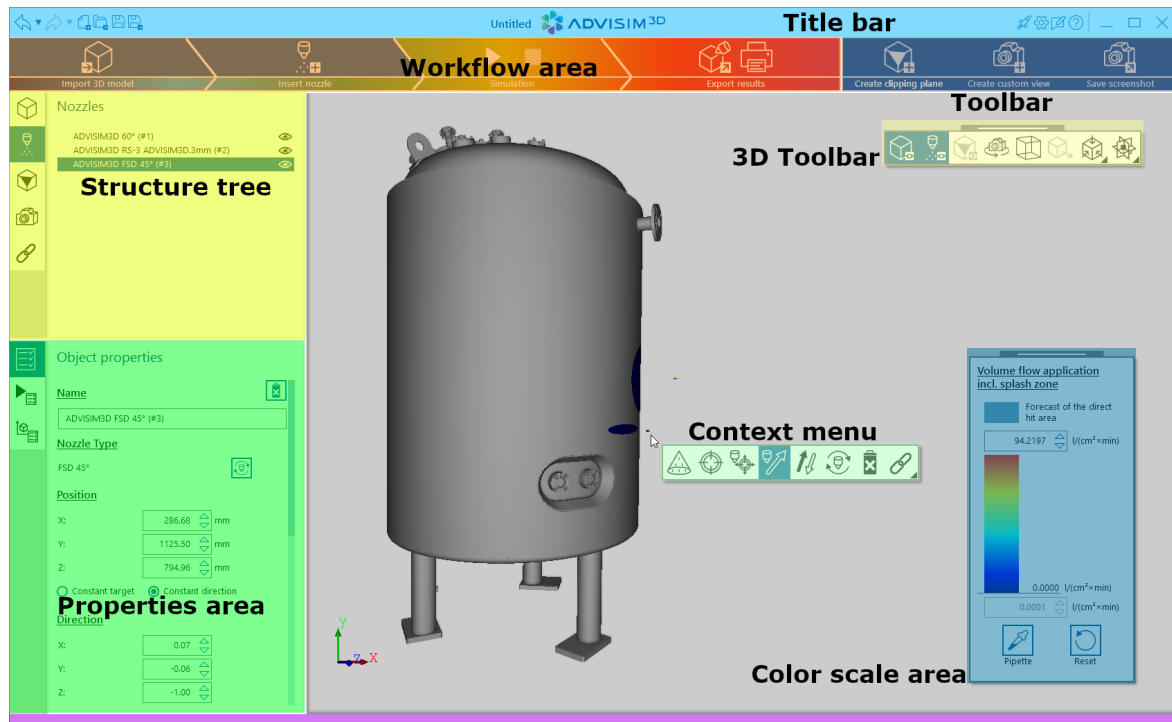
To contact us, please use the [contact form](#) on our [website](#).

## IV User documentation

This part contains information about the operating concepts and interactions with scene elements in ADVISIM<sup>3D</sup>.

### 4.1 Program structure

Here you get an overview of the structure of ADVISIM<sup>3D</sup>:



Information on the individual areas can be found here:

- [Structure tree](#)
- [Properties area](#)
- [3D-Toolbar](#)
- [Context menu](#)
- [Color scale area](#)

### 4.2 Quick Start Guide

The Quick Start Guide opens directly when starting ADVISIM<sup>3D</sup>. Alternatively, you can access the Quick Start Guide by clicking the Quick Start Guide button in the title bar (marked red in the image).





The Quick Start Guide offers you the possibility to go through the basic steps in ADVISIM<sup>3D</sup> step by step. The steps build on one another. For example, it is only possible to insert a nozzle if a model has been added before.

The Quick Start Guide offers the following points:

- [Import 3D model](#)
- [Insert nozzle](#)
- [Start simulation](#)
- [Export results / print report](#)

## 4.3 Release Notes

### Version 2.0.0 (23.02.2024)

#### New Features

- Customize existing spray balls or create your own with ADVISIM<sup>3D</sup> ([Customize and create new spray balls](#))
- Use of view-related clipping planes ([Clipping plane interaction](#))
- The following nozzle has been added to the nozzle catalog: ([Nozzle database](#))
  - "Alfa Laval - TE10B10001"
  - "Alfa Laval - TE10B13001"
  - "AWH - Tanko S30 LF 180° - 665 23 7600 2331"
  - "Guangzhou Cleaningspray Equipment - KLY-D20-03-0100-d20-TF"
  - "Kohiko - 280-8430-22-64-BS"
  - "Kohiko - 280-8431-21-72-DR"

- "Kohiko - 285-8260-22-212-DR"
- "Lechler - 632.644.30"
- "Lechler - 632.606.30.CC.00.0"
- "Lechler - 632.406.30.CC"
- "Lechler - 632.563.30.CC"
- "Lechler - 490.886.30.CG.00"
- "Lechler - 490.644.1Y.CC"
- "Lechler - 490.888.30.CG"
- "Lechler - 491.044.30.AM"

### Improvements

- Nozzle connection type is displayed in the nozzle catalog ([Nozzle database](#))
- Exclusive nozzles are supported ([Exclusive nozzles](#))
- Design / button icons have been revised

### Fixed Bugs

- Incorrect simulation of volume flow application for rotation cleaners and acute impact angles
- License information is not updated if the expiration date on the server has been changed

---

## Version 1.7.3 (28.08.2023)

### Improvements

- Color scale can represent volumetric flow values smaller than 0.001.

### Fixed Bugs

- Automatic adjustment of the minimum of the color scale after simulation results in almost no values being displayed in the color scale.

---

## Version 1.7.2 (12.05.2023)

### Improvements

- Automatic determination of connected model parts can be deactivated.

### Fixed Bugs

- Optimization of the grid prevents "flowing" around corners and curves

## Version 1.7.1 (31.01.2023)

### Improvements

- Updating the license agreement (EULA)

### Fixed Bugs

- Relocated project files with blanks in step file path cannot be loaded.
- Project files whose step file path no longer exists cannot be loaded.
- Project files are not upward compatible. The message when loading project files of newer versions is ambiguous.

---

## Version 1.7.0 (11.01.2023)

### New Features

- ADVISIM<sup>3D</sup> supports custom logos in the print report ([Custom Logo](#))
- The nozzle catalog has been extended by the following nozzles: ([Nozzle database](#))
  - "AWH Tanko S20"
  - "AWH Tanko S40"

### Improvements

- Simulation properties can be displayed in the print report ([Print report](#))
- Nozzle data import into ADVISIM<sup>3D</sup> online nozzle database was automated ([nozzle database](#))
- The following nozzles are imported release-independently for the first time:
  - "AWH Tanko S30"
  - "Lechler MicroSpinner 2"
  - "Lechler Micro Whirly"
  - "Lechler Pico Whirly"

### Fixed Bugs

- Black screen when simulating the ablated layer thickness. ([Simulation mode](#))
- Incorrect automatic adjustments of color scale after inserting a torrent cleaner ([Color scale](#))
- On high resolution monitors, the object properties obscure the coordinate system ([Coordinate system](#))
- Parameters start time, end time and duration of a motion have no units ([Motions](#))

## Version 1.6.0 (18.10.2022)

### New Features

- ADVISIM<sup>3D</sup> supports the definition of more than 6 clipping planes, where only 6 can be active at the same time ([Insert clipping plane](#))
- ADVISIM<sup>3D</sup> simulation kernel extended
  - Simulation of the ablated layer thickness within of 1m from all direct hit points ([Mode: Ablated layer thickness](#))
  - Modelling of the transition from non-decayed to the region of a decayed full beam by means of linear fades ([Workflow area of a solid stream](#))

### Improvements

- Help button has been added to the quick start guide and to the header of all dialog windows ([Quick start guide](#))
- Nozzle selection table can be sorted by all categories ([Nozzle selection dialog](#))
- License information and help are available in the menu bar ([Show license information](#))

### Fixed Bugs

- Screenshots and custom views in the print report do not have a time line and thus do not allow any conclusion about the simulation point in time. ([Take a screenshot](#))
  - Switching custom views does not switch the clipping planes ([Insert custom view](#))
- 

## Version 1.5.0 (25.04.2022)

### New Features

- ADVISIM<sup>3D</sup> supports the visualization of nozzles motion paths ([Show motion path](#))
- ADVISIM<sup>3D</sup> simulates motions even in the spray shadow-/preview ([Simulate motion](#))

### Improvements

- Added elements are indicated by a small blue dot in the structure tree

### Fixed Bugs

- Motions can be deactivated in the simulation result view.
  - Deactivated constraints can be activated in the simulation result view. They change the nozzle positions.
-

## Version 1.4.0 (28.03.2022)

### New Features

- ADVISIM<sup>3D</sup> supports rotating motions of individual nozzles ([Rotating motion](#))
- The following nozzle have been added to the nozzle catalog:
  - "Spraying Systems - MiniRokon - Tankjet D41800E - B3/8A4-13.6 EX"

### Improvements

- License information is loaded in the background so that ADVISIM<sup>3D</sup> remains continuously usable.
- The spray pattern completely covers the entire working area of the nozzle.
- The last simulation settings can be saved for new simulations.

### Fixed Bugs

- Changes to nozzle position with hidden shortcuts cannot be undone.
  - If links are in place and "Create new project" is clicked, "Delete link" is displayed in the list of actions that can be undone.
  - If a nozzle with animation is deleted from the project, the project cannot be simulated afterwards.
  - In case of a nozzle with motions and links, the action "Change nozzle type" results in the deletion of the motion/link.
  - If the selected model part surface is located behind a section plane, it is not highlighted when "Create link to model part surface" is selected.
- 

## Version 1.3.1 (14.01.2022)

### New Features

- ADVISIM<sup>3D</sup> supports linear movements of individual nozzles ([Linear motion](#))
- The following nozzles have been added to the nozzle catalog:
  - "AWH - Tanko-S30LF"
  - "Lechler - HygenicWhirly Baureihe 594"
  - "Lechler - Flachstrahldüse 660.486.17"
  - "Lechler - Flachstrahldüse 660.567.17"

### Improvements

- All notification windows have been adapted to the ADVISIM<sup>3D</sup> corporate design

### Fixed Bugs

- Wrong validations when entering values via spider elements.
  - No possibility to start the license dialog to enter another license when the maximum number of concurrently usable program instances is reached.
- 

## Version 1.2 (21.06.2021)

### New Features

- ADVISIM<sup>3D</sup> supports spray balls with variable spray angles up and down ([spray balls](#))
  - The following nozzles have been added to the nozzle catalog:
    - "AWH - TANKO-JM800-G13"
    - "AWH - TANKO-MX125-G14"
    - "AWH - TANKO-MX150-G12"
    - "AWH - TANKO-MX150-G14"
    - various sample nozzles for spray balls with variable spray angles.
- 

## Version 1.1 (16.04.2021)

### New Features

- Nozzle positioning via angles to the normals of the reference axes ([Angle to local reference planes](#))
- Coerce nozzle orientation after insertion of a constraint is activated by default ([Insert constraint](#))

### Improvements

- Minimum of the color scale also adjustable in preview ([Color scale area](#))
- Shortcut "Ctrl+O" for open project inserted ([Open project](#))
- Tooltips for window buttons
- Added "Hake nozzles" to nozzle catalog

### Fixed bugs

- Color scale minimum jumps when scrolling
  - Changing nozzle orientation with constant target leads to unexpected position change
  - Deselect nozzle with "ESC"
  - Installation problems on turkish speaking systems
- 

## Version 1.0 (15.10.2020) - ADVISIM<sup>3D</sup> is online

## New Features

- **ADVISIM<sup>3D</sup> goes online**
- Nozzle can be aligned to a model part axis or model part surface

## Improvements

- Simulation of the cleaning effect was activated again
- The help is now also available in English
- Properties area switches contextually

## Fixed bugs

- Deactivated logging is switched on at the next start
- Distance of the nozzle to the target point is not updated correctly

---

## Version 0.29 (11.06.2020)

### New Features

- New UX-Design including workflow bars, color scheme, layout, dialogs, icons
- Display of license expiration date refers to the remaining license usage period
- 9 more nozzles were published in the database

### Improvements

- Quick start guide can be displayed again after each work step
- Hidden nozzles are listed in the print report

### Fixed bugs

- Calculation of the volume flow distribution for solid streams is incorrect

---

## Version 0.28 (24.03.2020)

### New Features

- New user interface; SIMKOR is now called ADVISIM<sup>3D</sup>
- Conversion of a model part into nozzle possible
- Simulation results can be printed as a report

- Nozzle revision: Nozzles are automatically updated when the application is started

## Improvements

- Spray pattern simulation "upwards" of spray balls and surge cleaners is now even more realistic
- Overlay menus for controlling a 3D scene and for the color scale are now draggable and always visible

## Fixed Bugs

- Color scale cannot be saved as legend with screenshot

It is painful that we have temporarily abandoned the simulation of the cleaning effect, in which we have already invested enormous development efforts. But not everything that is desirable is also feasible and useful, so we have expanded this function for the time being in response to feedback from our test users. The prediction of the cleaning effect is so complex that the simulation would not yet provide realistic results.

---

## Version 0.27 (05.04.2019)

### New Features

- Simulation of spate cleaners is supported
- Changes on the color scale can now be [reset and restored](#)

### Improvements

- Slight design alterations, e.g. new icon
- Spray pattern simulation of spray balls optimized

### Fixed Bugs

- Flow rate application for spray balls does not scale correctly with the total flow rate
  - Reference distance for simulation is not saved
- 

## Version 0.26 (17.12.2018)

### New Features

- The set minimum and maximum of the color scales are saved in the project file



## Improvements

- Changing the nozzle type is possible via the context menu
- In the nozzle selection dialog, all nozzles from the cache are displayed by default z-axis of the model can be set as "top-direction"

## Fixed Bugs

- Maximum of the color scale is not saved at the first switching
- 

## Version 0.25 (10.10.2018)

### New Features

- Blocking of the currently opened project file, it can not be opened by other programs
- Simulations can be continued

### Improvements

- Performance beim Einfügen von Düsen wurde optimiert
- Current program version is displayed in the help window

### Fixed Bugs

- After switching the top direction, the view jumps
- 

## Version 0.24 (27.07.2018)

### New Features

- Models can be moved linearly to simulate a conveyor belt

### Improvements

- Displayed error messages were checked for comprehensibility and adapted
- Simulation component was further optimized
- Further progress in performance enhancement

### Fixed Bugs

- Drillings around the connection of a spray ball are ignored

- Spray patterns are distorted on straight edges
  - Spray nozzles show incorrect liquid distribution at edges
- 

## Version 0.23 (25.05.2018)

### New Features

- Nozzle database as a cloud service ensures high availability and flexibility
- New license management system supports the Concurrent licensing system

### Improvements

- Simulation component has been optimized

### Fixed Bugs

- Minor errors in the user interface, as well as in the help option to be reached in the software
  - Older SIMKOR projects with tank cleaners are not updated correctly
- 

## Version 0.21 (13.03.2018)

### New Features

- SIMKOR simulates the removed layer thickness in relation to a standard contamination ( $\mu\text{m}$ )
- SIMKOR simulates the amount of cleaning agent applied as direct volume application including footprint ( $\text{l}/\text{cm}^2$ )
- Realistic spray patterns reflect the pressure setting of a nozzle
- Pressure of a nozzle is adjustable

### Improvements

- Tanks are completely displayed up to a sight of 30 m
- Extension of the nozzle database with additional tank cleaners (After the model of the manufacturer Hake and Lechler)
- Adjustment of the default values at the start of the simulation to the nozzles used
- Minimum of the color scale after the simulation is 10% of the maximum

### Fixed Bugs

- Applied filter criteria do not match displayed filter criteria
- Correction of the rotational speed of Lecher-rotationally cleaner

---

## Version 0.20 (18.01.2018)

### New Features

- Intersections are displayed
- After stopping the simulation calculation, the previously calculated simulation steps are displayed in the results view

### Improvements

- Value setting via scrolling or rotating field (spinner) in the property area is applied directly to the positioning of the nozzles
- Specification of the minimum and maximum values of the color scale has been extended
- Significant increase in performance
- Tabs in the specification tree are fixed

### Fixed Bugs

- Minimum value of the simulation was displayed incorrectly
  - STEP files on network drives could not be opened via SIMKOR
- 

## Version 0.19 (28.11.2017)

### New Features

- Y and Z axes of the model can be set as the "top direction"
- Nozzles in the STEP-file can be converted into SIMKOR nozzles
- Model parts can be explicitly hidden for the simulation
- Detergent consumption is displayed for the entire simulation process as well as for individual nozzles in the result view

### Improvements

- Adaption of the installer to the region Austria

### Fixed Bugs

- Full jet nozzles radiate through other nozzles
  - Minor bugs related to undo redo and STEP import
-

## Version 0.18 (25.10.2017)

### New Features

- Round jet nozzles are now available in SIMKOR
- Own camera views can be created and saved
- Older project files update themselves automatically

### Improvements

- Extensive design customization of the area properties allows faster nozzle configuration, section planes, simulation parameters
  - Design customization of the nozzles and spray patterns stand out more clearly
  - Preset time values simplify the simulation
- 

## Version 0.17 (13.09.2017)

- Marketing-relevant adaptations within the framework of the industry and consumer trade fair Drinktec in Munich
- 

## Version 0.16 (08.09.2017)

### New Features

- New and innovative HELIX cleaner from the IVV has been implemented into SIMKOR
- Re-design of the entire SIMKOR surface
- Context menu for nozzles and section planes

### Improvements

- Simulated mode for the simulated cleaning effect has been refined
  - Performance improvement of the 3D scene as well as simulation
  - Triangulation has been improved so that the simulation result is better resolved
  - Redesign of the structure tree for greater clarity
  - Extension of the nozzle insert dialog for search criteria
  - Static nozzles now use the size specified in the database instead of a fixed size
- 

## Version 0.15 (22.06.2017)

## New Features

- Nozzles can be moved in time in the view of the simulation result
- Tank cleaners also rotate outside the center
- Ability to highlight a selected nozzle by spraying images
- Any selection of simulating modes

## 4.4 Projects

In ADVISIM<sup>3D</sup> projects user settings and scene elements are managed and can be saved and loaded locally.

### 4.4.1 Create new project

To create a new project in ADVISIM<sup>3D</sup>, select “Create new project” in the title bar.



This will result in the loss of all unsaved changes from the open project. A new project with the title “Untitled” is displayed.

### 4.4.2 Save project

To save a project, you have the following options:

- In the title bar select “Save project” to update an already saved project file. If no save path is specified yet, you can define it here.



- Under the menu item “Save project as” you can name a project file and save it in a folder.



- You can also save with `CTRL+S`.

Since a project file is locked as soon as it is opened by ADVISIM<sup>3D</sup>, it is only possible to overwrite a project file which is not opened by any other ADVISIM<sup>3D</sup> instance.

#### 4.4.2.1 Save simulation

If you have simulated a configuration, it is also possible to save the results. The results are stored in a subfolder next to the project file.

The name of the subfolder is composed as follows: `Projectname.Simulation/`

### 4.4.3 Open project

Saved project informations with the file extension `.smkr` can be restored via the menu item “Open project” in the title bar or with the shortcut `STRG + O`.



The following information is loaded:

1. The imported 3D model

- The 3D model is not stored itself, only its path. Note that the path information is stored as follows:
  - the absolute path to the 3D model,
  - the relative path from ADVISIM3D.exe to the 3D model
  - and the relative path from the project file to be loaded to the 3D model
- It is sufficient to find the model using one of these paths.

2. The visibility of the scene elements. This restores the structure tree with its shown and hidden parts.

- If the model file changes, the visibility information for the model is not restored.
- Model parts that were converted to nozzles are no longer available after loading.

3. The name and type of places nozzles as well their placement, orientation and visibility

4. Clipping planes with their name as well as their position, orientation and visibility

5. Custom view positions with their orientation and names.

6. The name, settings and visibility of the constraints.

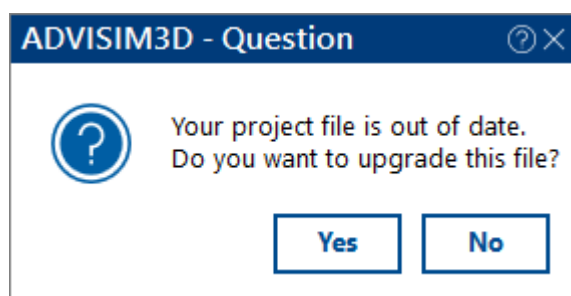
Please save all changes before loading a project, otherwise they will be discarded during loading process. Since a project file is locked as soon as it is opened by ADVISIM<sup>3D</sup>, it is only possible to open project files which are not opened by any other ADVISIM<sup>3D</sup> instance.

## Upgrade project

If you upgrade ADVISIM<sup>3D</sup>, the structure of project files may also change.

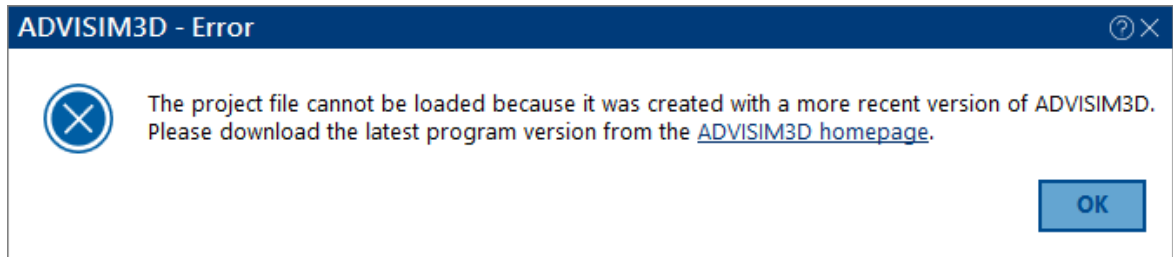
ADVISIM<sup>3D</sup> then offers you the possibility to update your projects.

Click "OK" to update the project. If you click "Cancel", the project remains as it is. It cannot be loaded with the new version of ADVISIM<sup>3D</sup>.



In some cases the project cannot be updated to the currently used ADVISIM<sup>3D</sup> version.

This is the case, for example, if the project comes from a newer ADVISIM<sup>3D</sup> version.



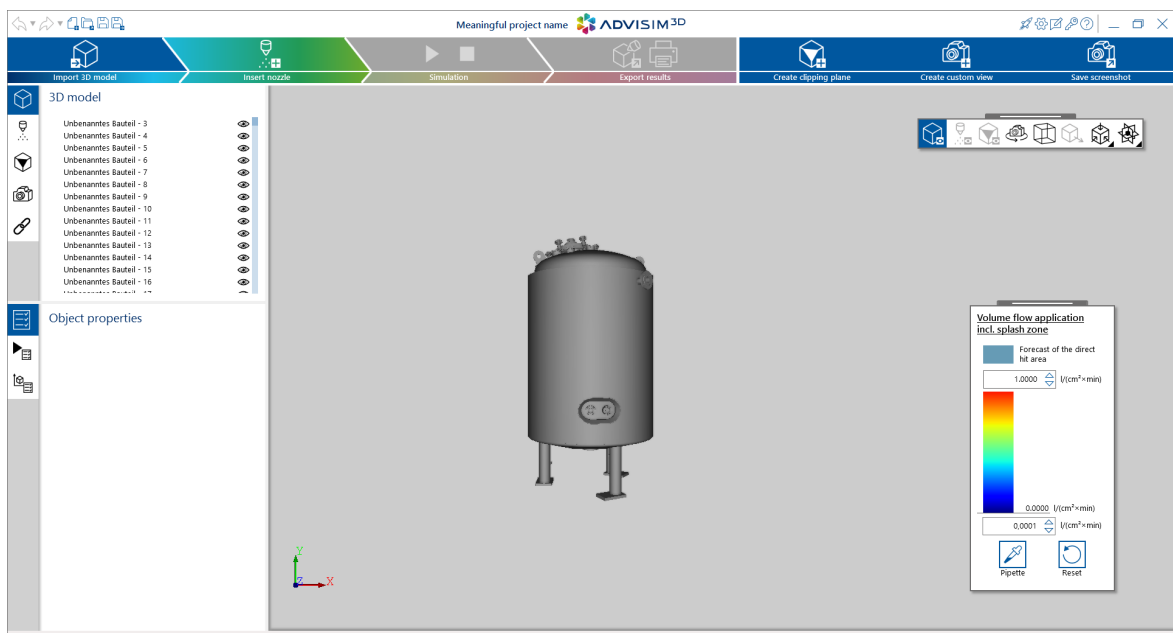
#### 4.4.3.1 Open simulation

If a project already receives simulation results, these are automatically loaded as well. When you finish the loading process, you will find yourself directly in the [results view of the simulation](#).

Note that [leaving the results view](#) will cause the results to be discarded.

## 4.5 View control

After you have [imported a model](#), it is displayed in the 3D component.



You now have the option to [rotate, move, and zoom in/out the object with the mouse](#).

You can also view the object from different [standardized angles](#).

After importing a model you have the possibility to interact with [nozzles](#) and [clipping planes](#).

You can also use the [structure tree](#) to highlight or hide individual scene elements in the left part of the screen.

The [properties area](#) contains information about a selected nozzle or clipping plane, constraint or custom view.

You can use the [color scale](#) to influence the display of the applied volume flow or the removed layer thickness

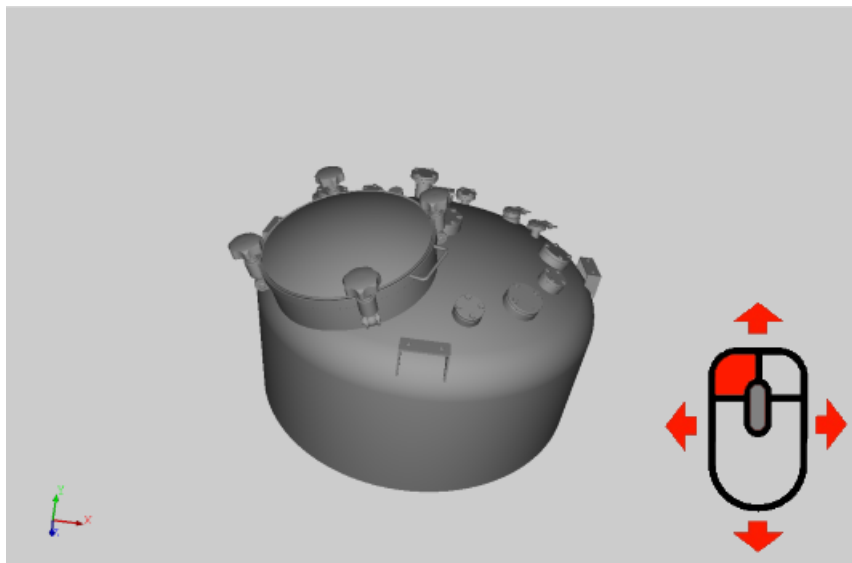
### 4.5.1 Mouse interaction

## Select object

To select an element in the 3D scene, left-click it with the mouse. It will then also be selected in the structure tree. With a mouse click on the background, all elements are deselected.

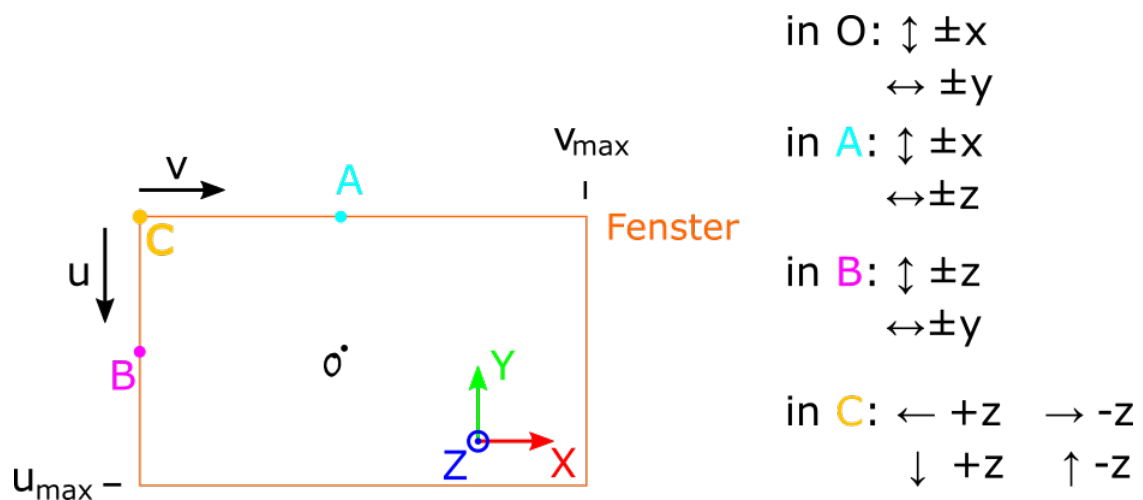
## Rotate

You can rotate the scene by positioning the mouse over the 3D component and then moving the mouse while holding down the left mouse button.



Since you can only move the mouse along two axes, but the 3D scene can be rotated around three axes, the following assignment is made:

When the mouse is in the middle of the 3D component, the scene rotates around the X and Y-axis.  
When the mouse is at the edge of the screen, it rotates around the Z-axis.



## Rotate camera



The [3D toolbar](#) contains the button for switching between model rotation and camera rotation.

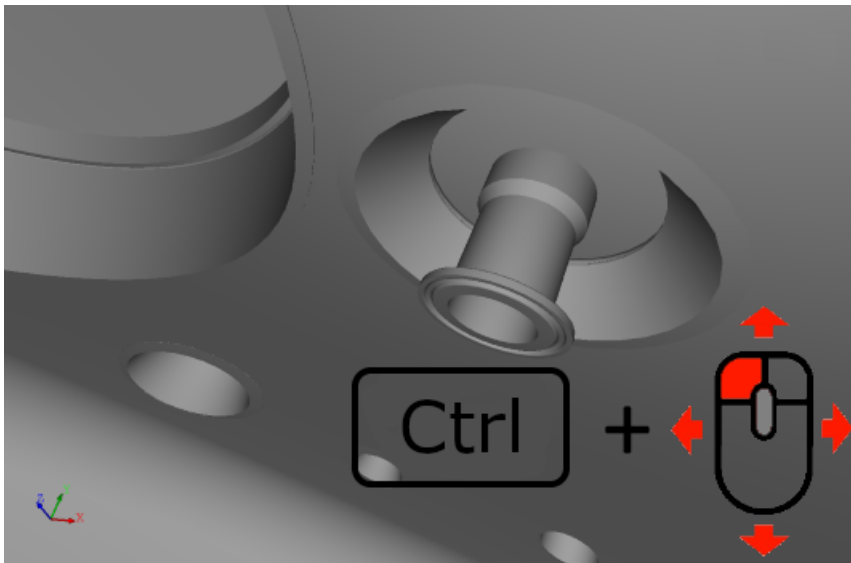


If it is active, the 3D scene will rotate around the point of view, like a cell phone camera panning around itself.

If it is inactive, the 3D scene rotates around the center of the model, like a carousel.

You can also switch between these modes by holding down the **CTRL** key.

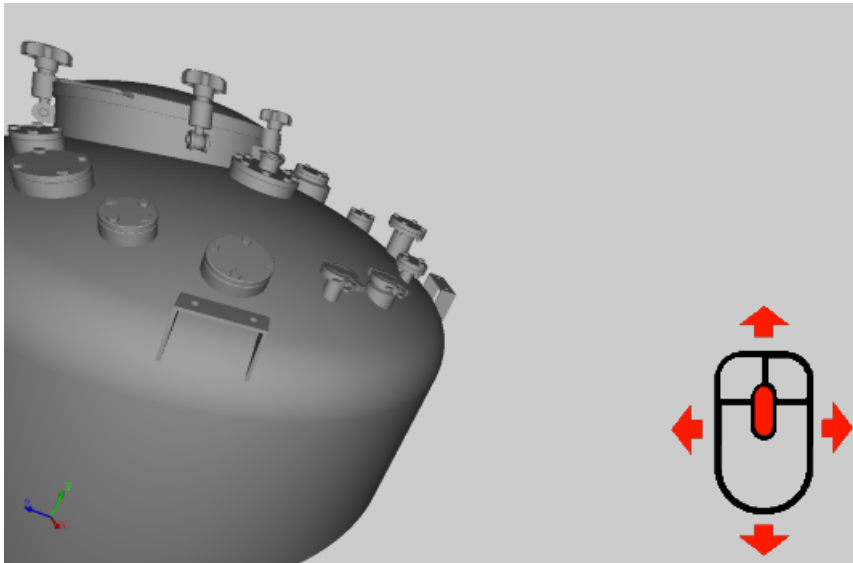
Hold down the **CTRL** key and then move the mouse while holding down the left mouse button.



## Move

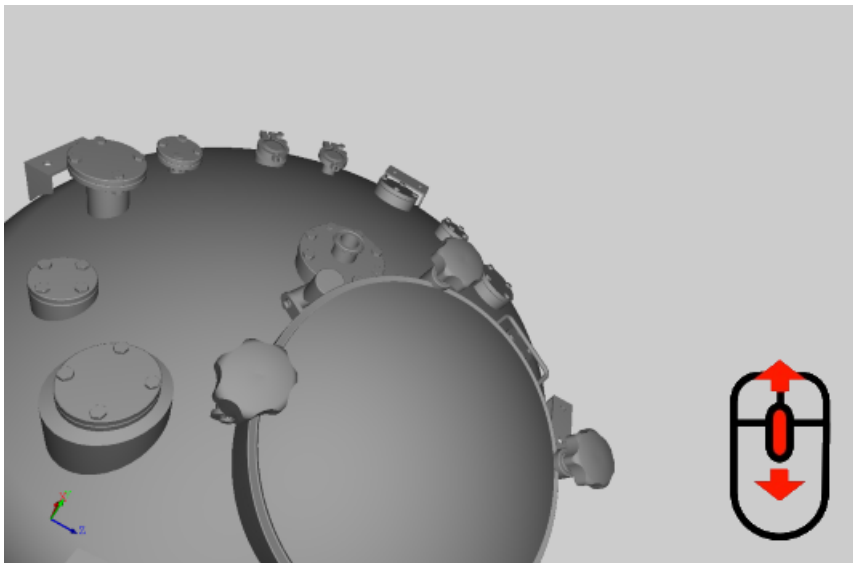
You can move the model by placing the mouse over the 3D component and then moving the mouse while holding down the middle mouse button.

*Note:* In some case the middle button is the mouse wheel itself.



## Zoom in/out

You can zoom in and out of the scene by positioning the mouse over the 3D component and then **rotating the mouse wheel**. The mouse cursor represents the target point of the zoom during this process.



## Deselect elements

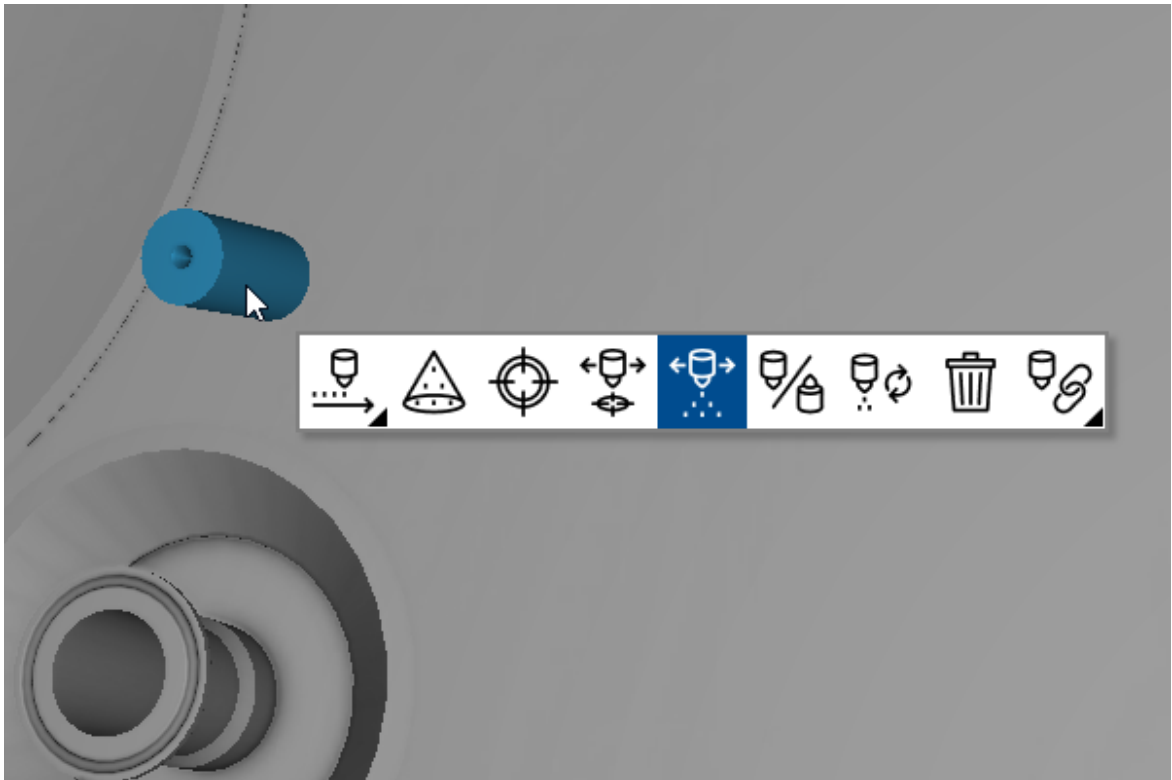
To deselect a selected object (model part, nozzle or clipping plane), press the **ESC** key.

### 4.5.2 Open context menu

ADVISIM<sup>3D</sup> offers the possibility to open a context menu by right clicking in the 3D scene. This shows a different selection of actions, depending on which action preceded it.

To open the context menu, right-click on a scene element (see picture). You can also open the context menu by right-click on a scene element in the [structure tree](#).

Modes like [inserting a nozzle](#), also have their own context menu actions.



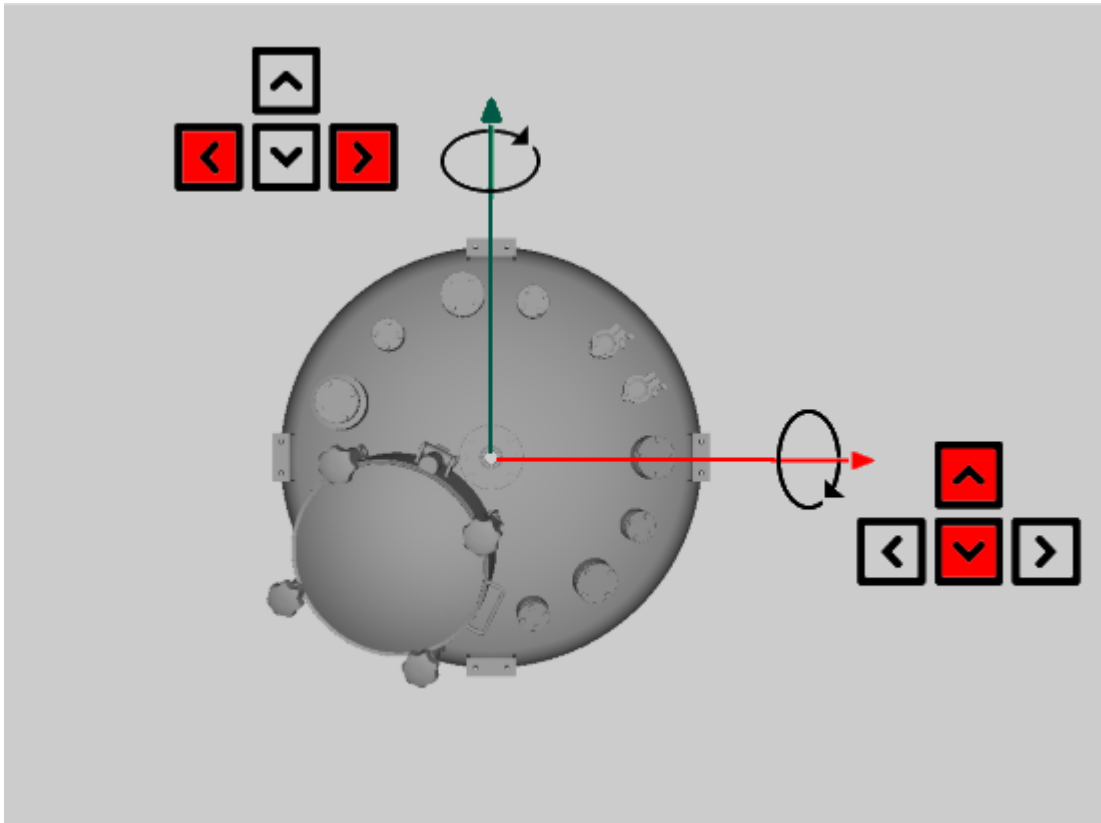
You can access the following functions via the respective context menus:

- **Model with at least two components:**
  - [Convert one of the model parts into a nozzle](#)
- **Clipping plane:**
  - [Invert direction](#)
  - [Delete clipping plane](#)
- **Insert nozzle-mode:**
  - [Insert nozzle in viewing direction](#)
  - [Insert nozzle along surface normal](#)
- **Selected nozzle:**
  - [Workspace on / off](#)
  - [Set new target point](#)
  - [Move nozzle with constant target point](#)
  - [Move nozzle with constant direction](#)
  - [Invert nozzle direction](#)
  - [Change nozzle type](#)
  - [Delete nozzle](#)
  - [Insert nozzle constraint](#)

### 4.5.3 Keyboard interaction

## Rotate

To rotate the scene left and right, up and down, you can use the corresponding arrow keys on the keyboard. With the left mouse button you can click on the 3D component, which is then selected for keyboard control.

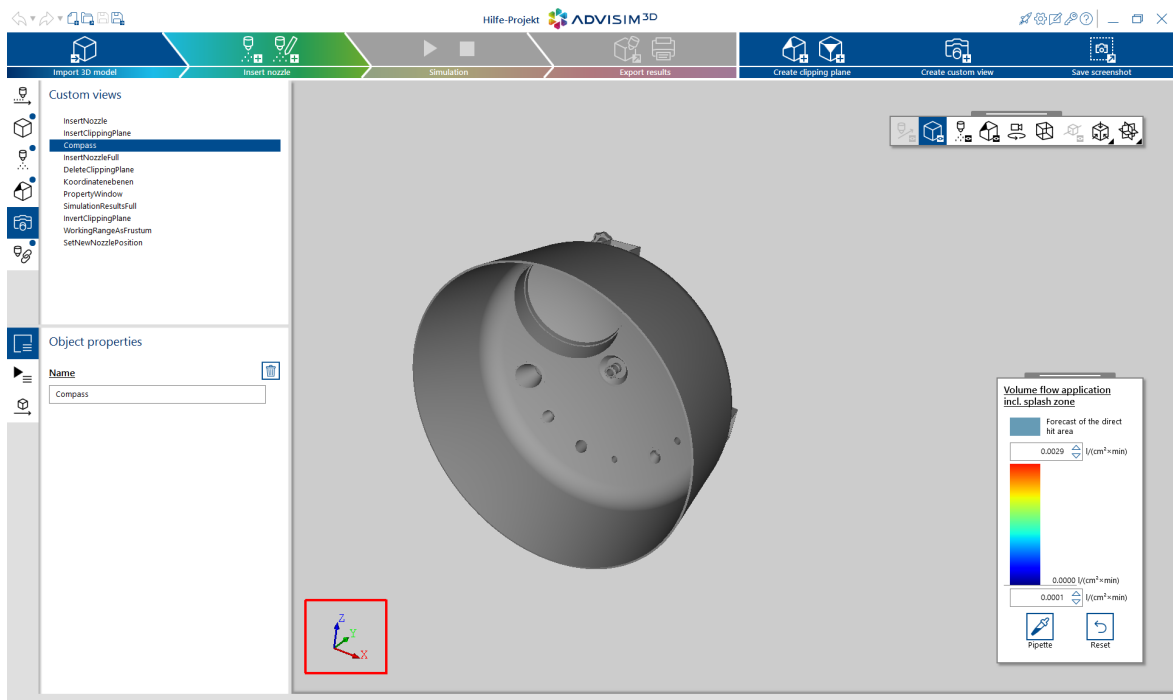


## Deselect elements

To deselect a selected object (model part, nozzle or clipping plane), press the **ESC** key.

### 4.5.4 3D coordinate system

By default ADVISIM<sup>3D</sup> shows you the current orientation of a scene in the 3D coordinate system. When the camera moves, the view changes.

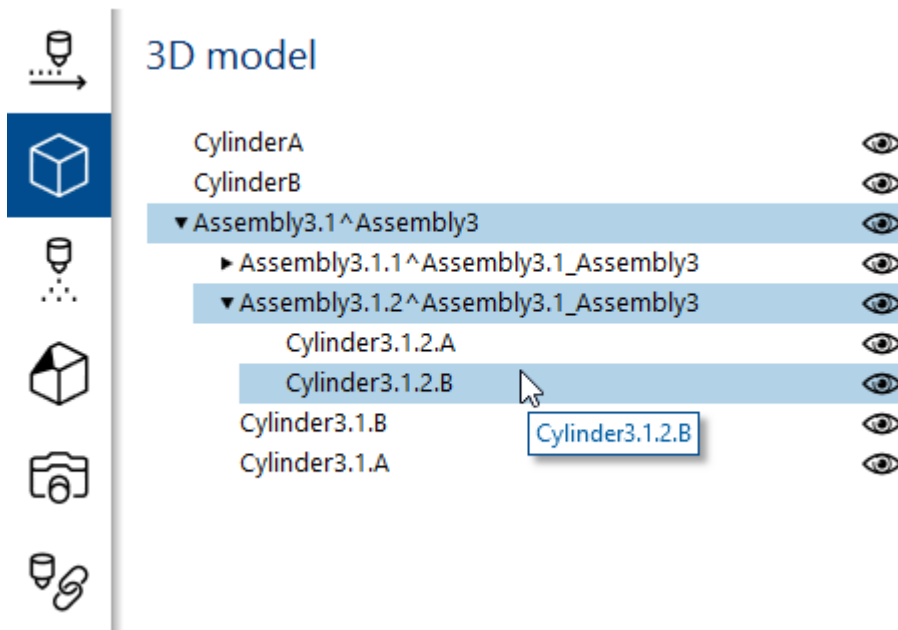


#### 4.5.5 Structure tree



The left program area below the workflow area is called the structure tree and represents the components of the scene.

For each category a tab is displayed to structure the corresponding elements:

- motions
- model parts
- nozzles
- clipping planes
- custom views
- constraints



When selecting a model part in the structure tree, the selected element is highlighted in blue in the 3D view. For assemblies, the corresponding sub-elements are also highlighted.

The symbols behind the scene element stand for shown  or hidden . By clicking on the eye, you can also switch from show to hide and vice versa.

If a scene element has sub-elements, hiding/showing the parent element also affects the sub-elements. Hidden elements no longer influence the simulation result.

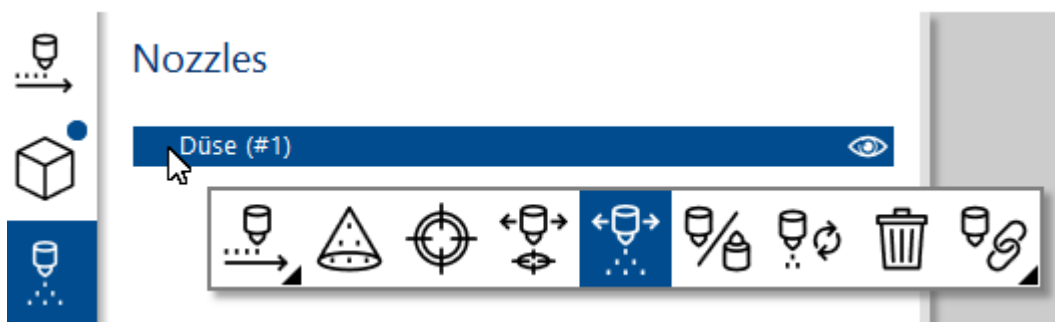
[All scene elements of a category can be shown and hidden in the 3D toolbar.](#)

After you have created a nozzle in the 3D scene, it is displayed in the structure tree in the “Nozzles” tab. Nozzles are [shown and hidden](#) in the same way as model parts.

Clipping planes added to the scene are displayed in the structure tree in the “clipping planes” tab. Clipping planes can be [shown and hidden](#) like all scene elements.

There are also separate tabs for motions, custom views and constraints. Custom views cannot be hidden.

By right-clicking on a specific scene element, the respective [context menu](#) can be opened via the structure tree:



### 4.5.6 Simulation views

In the properties view of the simulation there are two view modes you can switch between.

#### Simulation properties

On top

Y Axis ▼

Frames per second: 2

Time to simulate: 0:03:00 (hh:mm:ss)

#### Simulation mode



Direct volume application  
incl. footprint



Ablated layer thickness

#### Fluid consumption in litres

Nozzle description	Current	Total
ADVISIM3D 60° (#1)	3.0	3.0
All nozzles	3.0	3.0



- [Direct volume application including footprint](#)



- [Ablated layer thickness](#)

**Caution!** – The result of the ablated layer thickness must be activated in the [settings](#), as it is deactivated by default.

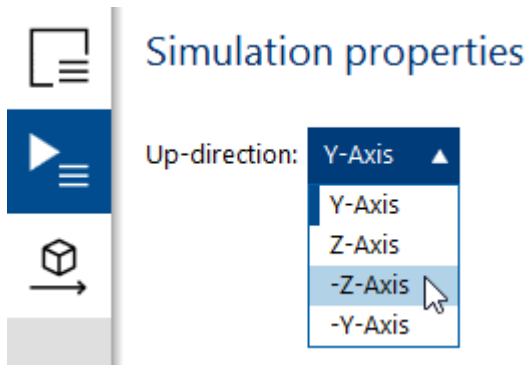
The results show an estimation for the expected cleaning process under the ideal conditions of a laboratory test.

The results serve exclusively as a basis for interpretation by an experienced engineer under consideration of the actual cleaning conditions and alone do not provide a valid statement about the expected layer ablation.

### 4.5.7 Up-direction of the model

In order for the model to be displayed as expected and used accordingly in the [calculation of the simulation results](#), it is possible to change the up-direction of the model.

Some CAD programs use different coordinate systems which can lead to problems with the up-direction.



To do so, select the desired up-direction in the properties area under simulation properties:

- Y axis
- Z axis
- -Z axis

This also changes the corresponding display of the [standard views](#). By default, the Y-axis is assumed as the up-direction.

The up-direction is saved when the project file is saved.

#### 4.5.8 Undo/Redo actions

In ADVISIM<sup>3D</sup> you have the possibility to undo or redo actions that change the project file.

For this purpose there are the two arrow buttons in the title bar of ADVISIM<sup>3D</sup>.

The tooltip shows you which action is undone.



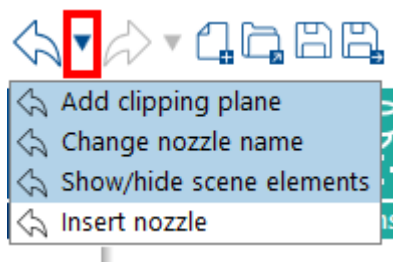
Click the button or enter the designated key combination to trigger the action.

“Undo” resets the last action. “Redo” restores the last undone action.

When you load a project, create a new project or import a new STEP model, you can neither undo nor repeat previous actions.

A click on the small arrow opens a list of actions (see picture), which can all be undone or redone.

Move the mouse to the corresponding entry and click the left mouse button. All actions that are highlighted are simultaneously undone or repeated by left-clicking with the mouse.



##### Notes:

- If you switch to or leave the simulation results mode, all previous actions cannot be undone or redone.



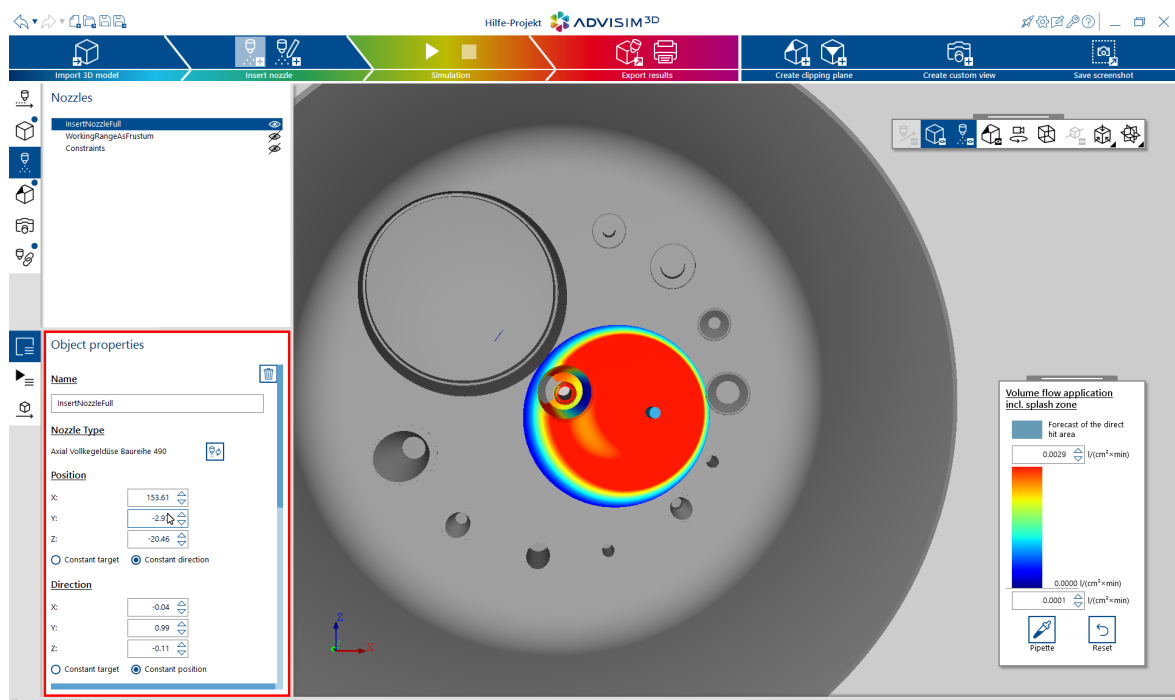
- When you move a nozzle or clipping plane, all actions that occur in a period of less than 750 milliseconds from the last action are combined in one undo / redo step.
- Similar to other programs, you can use the shortcut **CTRL+Z** (for undo) or **CTRL+Y** (for redo).

## 4.6 Properties area

Information about a scene object is clearly displayed in the properties area. Furthermore, you have the possibility to redefine some properties.

Depending on the type of the selected object, different information can be displayed and edited.

If a nozzle, clipping plane, custom view or constraint is selected, its properties are displayed in the properties area. In the simulation results view you can also view the properties of the simulation.



The properties area has several groups that indicate that the corresponding property is editable.

To confirm an entry, either press the **ENTER** key or switch to another field.

An incorrect entry is indicated by the **!**-icon. Move the mouse over it to see the reason for the error.



### 4.6.1 Properties area for nozzles

The properties area for nozzles provides information and tools that can be used to precisely locate or position the nozzle in space.

## Object properties

**Name**

ADVISIM3D Beispieldüsen Vollkegeldüse 45° fiktiv (#1)

**Nozzle Type**

Vollkegeldüse 45°

**Position**

X:

253.28

mm

Y:

253.72

mm

Z:

840.82

mm

☐ Constant target
 ☒ Constant direction

**Direction**

X:

0.01

Y:

0.01

Z:

-1.00

☐ Constant target
 ☒ Constant position

**Angle to the normal of local reference planes**

Main axis:

179

°

Minor axis:

-1

°

YX

XZ

YZ

**Working pressure**

Pressure range:

0.5 bar - 5 bar

Current pressure:

2.0

bar

**Up-Vector**

Angle:

0

°

**Distance**

340.86

mm

**Information**

Id:	1
Description:	Vollkegeldüse 45°
Manufacturer:	ADVISIM3D Beispieldüsen
Item number:	fiktiv
max. working range:	1 mm - 750 mm
Height:	20 mm
Diameter:	8 mm
Spraying angle:	45.0°
Volume flow:	1.00 l/min

The area can be divided into 9 parts, of which the first 8 can be changed.

- [Name of nozzle](#)
- [Type of nozzle](#)
- [Position of nozzle](#)
- [Orientation of nozzle](#)
- [Angle to local reference planes](#)
- [Working pressure of a nozzle](#)
- [Up-vector of a nozzle](#)
- [Distance of a nozzle to the target](#)
- [Properties area](#)

## Deleting a nozzle

A nozzle can be deleted by clicking the -button in the name area of the nozzle.

### 4.6.1.1 Change names

When the input is confirmed, the name is updated and displayed in the project's structure tree.

Name 

Zielstrebige Düse

An empty name is not valid and will not be applied.

## Deleting a nozzle

A nozzle can be deleted by clicking the -button in the upper right area


### 4.6.1.2 Changing the type

In this area the type of the selected nozzle is displayed. Additionally you have the possibility to change the nozzle type.

Nozzle Type

60° Test 

## Changing the nozzle type

If you want to change the type of a selected nozzle, click on the -button. This opens the [nozzle selection dialog](#). Here you can select and confirm a new nozzle type. Changing the nozzle type affects the entered object properties as follows:

- **Name:** If the name of the nozzle was previously adapted, it is retained. Otherwise the name is adapted to the new type.
- **Position:** Values are retained.
- **Orientation:** Values are retained.
- **Working pressure:** If the previously entered pressure is applicable [to the pressure](#) range of the new nozzle type, it is applied. Otherwise the current pressure is reset to the minimum.
- **Up-vector:** Value is retained.
- **Distance:** Value is retained.
- **Information:** Values are adapted to the new nozzle type.

#### 4.6.1.3 Change position

You can adjust the position of a nozzle by changing the three axes.

Furthermore, you can choose between two modes for moving:

- **Constant target:** The target point of the nozzle remains the same, the orientation of the nozzle changes.
- **Constant direction:** The direction remains, but the target point changes.

##### Position

X:	888.54	▲ ▼
Y:	406.68	▲ ▼
Z:	230.37	▲ ▼

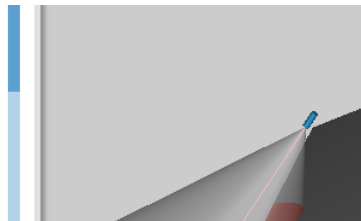
☐ Constant target ☒ Constant direction

If a nozzle has no direct target, it cannot be moved with a constant target. In this case the corresponding option is disabled.

Position

X:	90.85	▲ ▼
Y:	160.92	▲ ▼
Z:	592.19	▲ ▼

☐ Constant target ☒ Constant direction



#### 4.6.1.4 Change direction

Changing the direction behaves adequately to change the nozzle position.

However, the spray direction is changed instead of the position.

Here there are also two modes for changing the nozzle direction:

- **Constant target:** The target point of the nozzle remains the same. The nozzle is turned around the target point.
- **Constant position:** The position remains constant and the target position is changed.

##### Direction



X:	-1.00	▲ ▼
Y:	0.00	▲ ▼
Z:	0.00	▲ ▼



☐ Constant target ☒ Constant position



**Note:** The zero vector cannot be entered. Furthermore all direction vectors are normalized. Since the top vector must always be orthogonal to the direction, it is calculated and updated.

If the nozzle has no direct target, it is not possible to align the nozzle with a constant target. In this case the corresponding option is disabled.

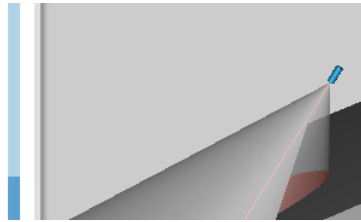
Direction

X:   

Y:   

Z:   

☐ Constant target ☒ Constant position



#### 4.6.1.5 Angle to the normal of local reference planes

In this area you can set the rotation of the nozzle according to the normals of the reference planes. Enter the rotation as an angle in degrees.



You can choose between one of the three reference planes, which is the reference plane for the rotation. If you click on another reference plane, the rotation with respect to it is recalculated.



The main axis refers to the first specified axis around which rotation takes place.

The minor axis refers to the second specified axis around which rotation takes place.

Only angles between  $-180^\circ$  and  $180^\circ$  are allowed as input.

#### Angle to the normal of local reference planes

Main axis:    °

Minor axis:    °



**Note:** The selected reference plane is saved in the project.

#### 4.6.1.6 Change working pressure

In ADVISIM<sup>3D</sup> you have the possibility to regulate the applied operating pressure of the nozzle. The pressure has a direct influence on the volume flow in a simulation.


#### Working pressure


Pressure range: 2 bar - 6 bar

Current pressure:    bar

**Note:** The working pressure of the nozzle has no influence on the working range covered by the nozzle. This is constant.

#### 4.6.1.7 Change up-vector

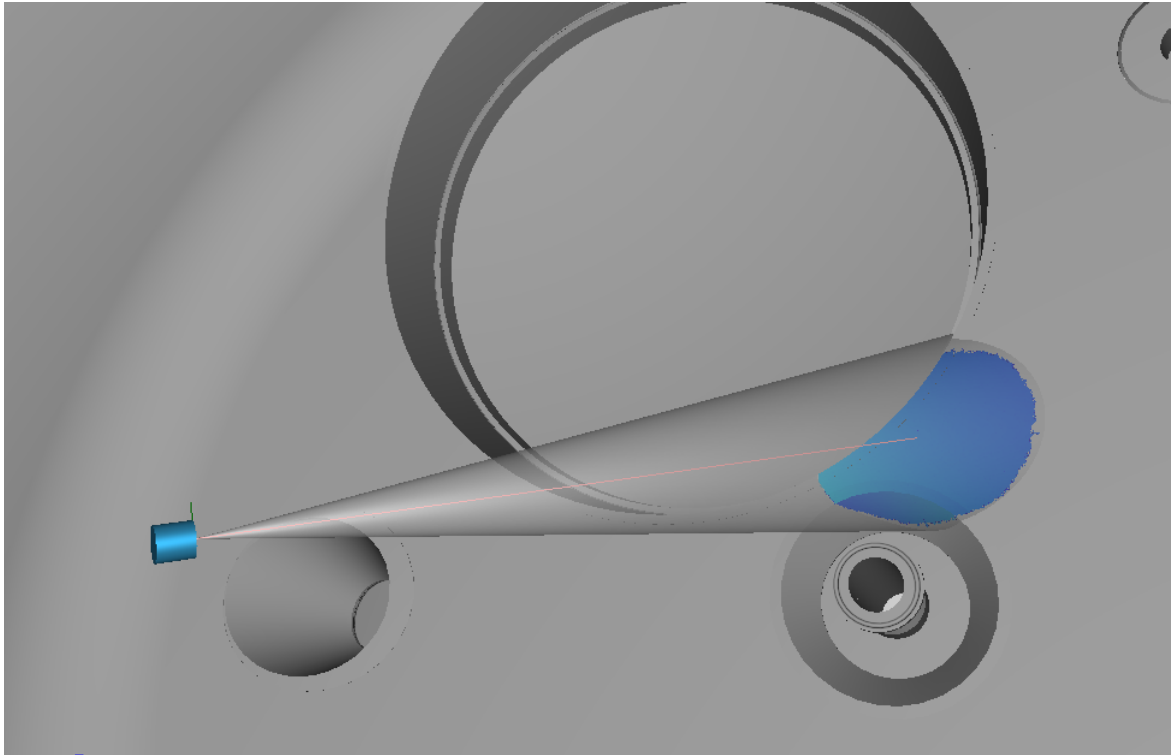
With the  -button you can rotate the nozzle counterclockwise.

With the  -button you can rotate the nozzle clockwise.

Each click rotates by 1°. If you click and hold one of the two buttons longer, the action will continue to be executed. You can rotate the nozzle from -359° to 359°. A value greater than 0 corresponds to the mathematically positive direction of rotation and a value less than 0 corresponds to the mathematically negative direction of rotation.

### Up-Vector

Angle:

 °

#### 4.6.1.8 Change distance

This value specifies the distance of the nozzle focus beam between the nozzle opening and the first point of impact on the 3D object.

### Distance

 mm

If the nozzle has no target, no value can be entered. This is indicated by a red warning triangle.

### Distance

  mm

#### 4.6.1.9 General properties

Here you get general information about the nozzle, which are also displayed in the nozzle selection dialog.

### Information

Id: 2  
Description: 60° Test  
Manufacturer: SIMKOR  
Item number:  
max. working range: 1 mm - 725 mm  
Height: 19 mm  
Diameter: 8 mm  
Spraying angle: 60.0°  
Volume flow: 1.30 l/min

If no spraying angle is specified the parameter is not displayed.

## 4.6.2 Clipping plane properties area

The clipping plane properties area provides information and tools to change the properties of the clipping plane.

### Object properties

#### Name



Section plane1

#### Position

X: 0.00 mm  
Y: 15.75 mm  
Z: 0.00 mm

#### Normal

X: -0.90  
Y: -0.40  
Z: -0.18

#### Angle to local reference planes

Main axis: -101 °  
Minor axis: 24 °



#### Information

Id: 1

The area can be divided into 5 parts, of which the first 3 can be edited.

- [Name of the clipping plane](#)
- [Position of the clipping plane](#)
- [Clipping plane normal](#)
- [Angle to local reference planes](#)
- [ID of the clipping plane](#)

## Delete a clipping plane

A clipping plane is deleted by clicking the -button in the name area of the clipping plane.


### 4.6.2.1 Change names

In the editing mode for the clipping plane name, the name can be reassigned.  
After confirming the input, the name is updated and displayed in the project's structure tree.

Name 

Section plane1







## Delete a clipping plane

is deleted by clicking the -buttons.

### 4.6.2.2 Change position

In the change mode for the position, you can locate the exact center of the clipping plane.

Position







X:	<div>0.00  </div>	mm
Y:	<div>15.75  </div>	mm
Z:	<div>0.00  </div>	mm

### 4.6.2.3 Change normal

Here you can change the normal of the clipping plane.  
The clipping plane hides the part of the model that is opposite to its plane normal.



### Normal

X:	<input type="text" value="-0.90"/>	 
Y:	<input type="text" value="-0.40"/>	 
Z:	<input type="text" value="-0.18"/>	 

**Note:** The zero vector is not allowed. The orientation of the clipping plane is normalized after the input.

#### 4.6.2.4 Angle to local reference planes

In this area you can set the rotation of the clipping plane to reference planes. Enter the rotation as an angle in degrees.





You can choose between one of the 3 reference planes, which is the reference plane for the rotation. If you click on another reference plane, the rotation with respect to it is recalculated.

The main axis refers to the first specified axis around which rotation takes place.

The minor axis refers to the second specified axis around which rotation takes place.

Only angles between  $-180^\circ$  and  $180^\circ$  are allowed as input.

### Angle to local reference planes

Main axis:	<input type="text" value="-101"/>	 	°	<div><div>YX</div><div>XZ</div><div>YZ</div></div>
Minor axis:	<input type="text" value="24"/>	 	°	

**Note:** The selected reference plane is saved in the project.

#### 4.6.2.5 General clipping plane properties

The ID of the clipping plane is specified here. It clearly identifies the clipping plane in the project.

### Information

Id: 1

#### 4.6.3 Custom view properties area

The properties area for custom views offers the possibility to edit the name of a custom view or delete custom views.

## Object properties

### Name




Custom view 1

## Edit the name of a custom view

In the edit mode for custom views you can also assign a new name.

When the input is confirmed, the name is updated and displayed in the project's structure tree.

## Delete a custom view

You can delete a custom view by clicking the -buttons.

### 4.6.4 Constraints properties area

The constraints properties area provides information and tools to change the properties of the constraint or to delete a constraint. Depending on the type of constraint used, the actual structure or the setting options change.

## Constraint parallel to a model part plane

### Object properties

#### Name



Along axis of modelpart (nozzle #2)

#### Alignment

☐ Align nozzle direction to axis

#### Information

Id: 1  
Constraint for nozzle: 2

The area can be divided into 3 parts, of which the 2nd can be edited.

- [Name of constraint](#)

- [Orientation of constraint](#)
- [Properties area](#)

## Constraint along a model part axis

### Object properties

#### Name





Parallel to model part plane (nozzle #3)

#### Alignment

☐ Align nozzle direction vertical to model part plane

#### Distance

0.00   mm

#### Information

Id: 1


Constraint for nozzle: 3

The area can be divided into 4 parts, of which the 2nd and 3rd can be edited.

- [Name of constraint](#)
- [Orientation of constraint](#)
- [Distance of constraint](#)
- [Properties area](#)

## Delete a constraint



A created constraint can be deleted by clicking the -button in the name area of the constraint.

### 4.6.4.1 Show name

This area displays the name of the selected constraint.

The name of the constraint is automatically generated by ADVISIM<sup>3D</sup> and cannot be edited. It consists of the type of constraint and the associated nozzle number. In this area you have the possibility to delete the selected constraint.

Name



Parallel to model part plane (nozzle #3)

## Delete a constraint



A selected constraint can be deleted by clicking the -button in the upper right area.

### 4.6.4.2 Change orientation

In this area you have the possibility to limit the freedom of movement of a nozzle.

## Constraint: Parallel to a model part plane

When this option is selected, the associated nozzle automatically aligns itself perpendicular to the associated model part plane. With this setting, the orientation of the nozzle within the nozzle properties area can no longer be edited. If you remove the orientation again, you are able to freely process the nozzle orientation again.

### Alignment

☐ Align nozzle direction vertical to model part plane

## Constraint: Along a model part axis

When this option is selected, the corresponding nozzle automatically aligns itself along the corresponding part axis.

### Alignment

☐ Align nozzle direction to axis



## Effects on the nozzle properties area

As a result, the [orientation of the nozzle cannot be edited within the nozzle properties area](#). If you remove the orientation again, you are able to freely process the nozzle orientation again.

### 4.6.4.3 Change distance

The distance indicates the distance of the nozzle focus stream to the first point of impact of the associated model part plane. The entered value must be positive.

### Distance

0.00   mm

#### 4.6.4.4 General properties

This view lists the general information of a constraint.

### Information

Id: 1

Constraint for nozzle: 3

#### 4.6.5 Simulation properties area

The structure of the simulation properties area changes depending on whether it is pulled up before or after the simulation.

### Properties area before the simulation

Before the simulation, the desired axis of the upper view can be selected in the simulation properties.

On top

Y Axis 

### Properties area after the simulation

After running the simulation, the simulation property information will expand. The [top direction](#) can no longer be edited at this point and is only displayed.

Additional parameters of the simulation (interval and duration) are now displayed.

Also the liquid consumption over the duration of the simulation of all nozzles used in the simulation is displayed in detail and in total.

On top Y Axis  
Frames per second: 1  
Time to simulate: 0:05:00 (hh:mm:ss)

Fluid consumption in litres

Nozzle description	Current	Total
ADVISIM3D 60° (#1)	5.0	5.0
Lechler Sprühkugel Baureihe 591 591.A21.17 (#2)	905.0	905.0
ADVISIM3D RS-3 ADVISIM3D.3mm (#3)	33.3	33.3
All nozzles	943.3	943.3

## 4.7 Toolbar

In the subchapters you will find information on:

- [Insert clipping plane](#)
- [Insert custom view](#)
- [Take a screenshot](#)

### 4.7.1 Take a screenshot

The toolbar contains the button to create and save an image of the 3D scene.



This is only active when a [model has been loaded into the 3D scene](#). When you click the button, a save dialog opens and the current image of the 3D scene can be saved as a PNG file.

In each case, the current 3D scene is saved without the structure tree and properties area. Timelines and color scales are retained.

## 4.8 3D toolbar

The 3D toolbar offers you various options to influence the display of the 3D scene.



It provides the following options:

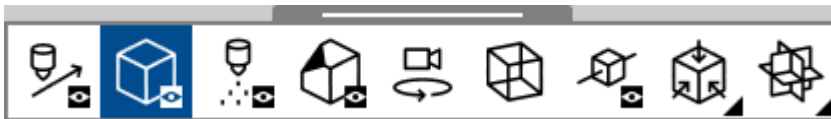
- [Show motion path](#)
- [Show/hide all model components](#)
- [Show/hide all nozzles](#)
- [Show/hide all clipping planes](#)
- [Toggle between scene rotation \(default setting\) and interior view](#)

- [Switch between shaded rendering mode \(default setting\) and wireframe rendering mode](#)
- [Show direction of movement](#)
- [Selection of a standard view](#)
- [Show coordinate planes](#)

#### 4.8.1 Set visibility of all scene elements

All scene elements of a category can be shown or hidden in the toolbar. Previously hidden elements are also displayed again.

##### Show and hide model parts



Note that at least one model part must be shown to start the simulation.

##### Show and hide nozzles



Note that at least one nozzle must be shown to start the simulation

##### Show and hide clipping planes

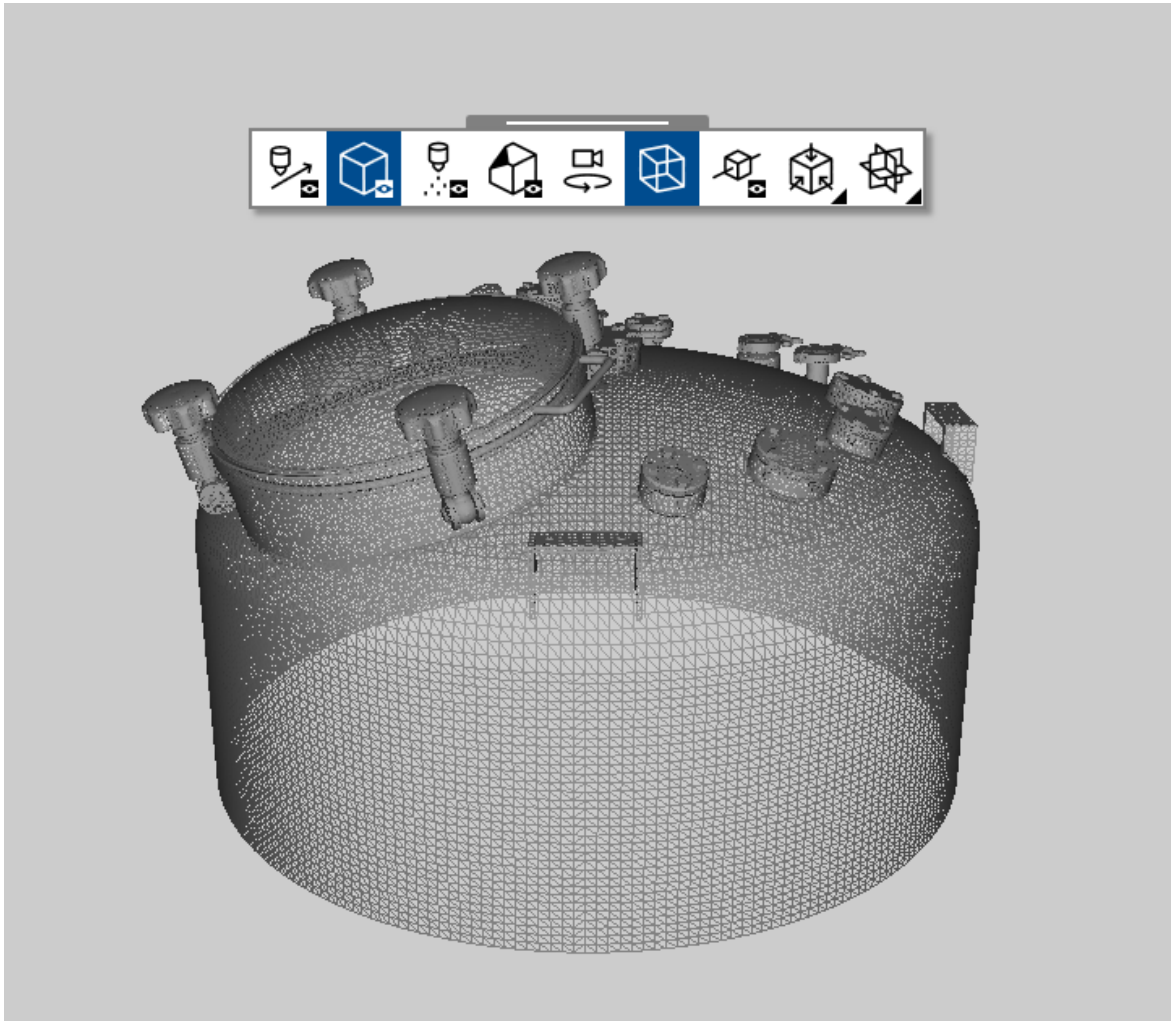


#### 4.8.2 Switch view mode

You can switch between the normal view with the surfaces of the model and the wireframe rendering mode.



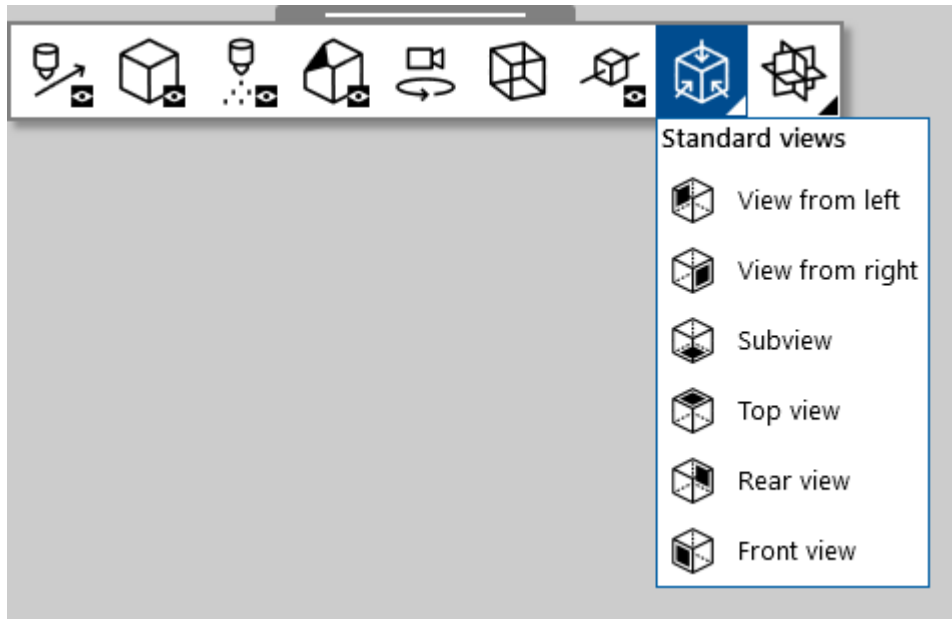
When you activate the button, the wireframe model is displayed. If you click the button again, the button becomes deactivated and the surfaces of the model are displayed again.



### 4.8.3 Standard views

You have the option of viewing the scene along the coordinate axes. These six views are called standard views and can be selected in the [3D toolbar](#).





The black side of the cube indicates the side from which the scene is to be viewed.

When the model is opened for the first time, it is displayed in front view. When using the standard views, the current movement and zoom of the scene are reset.

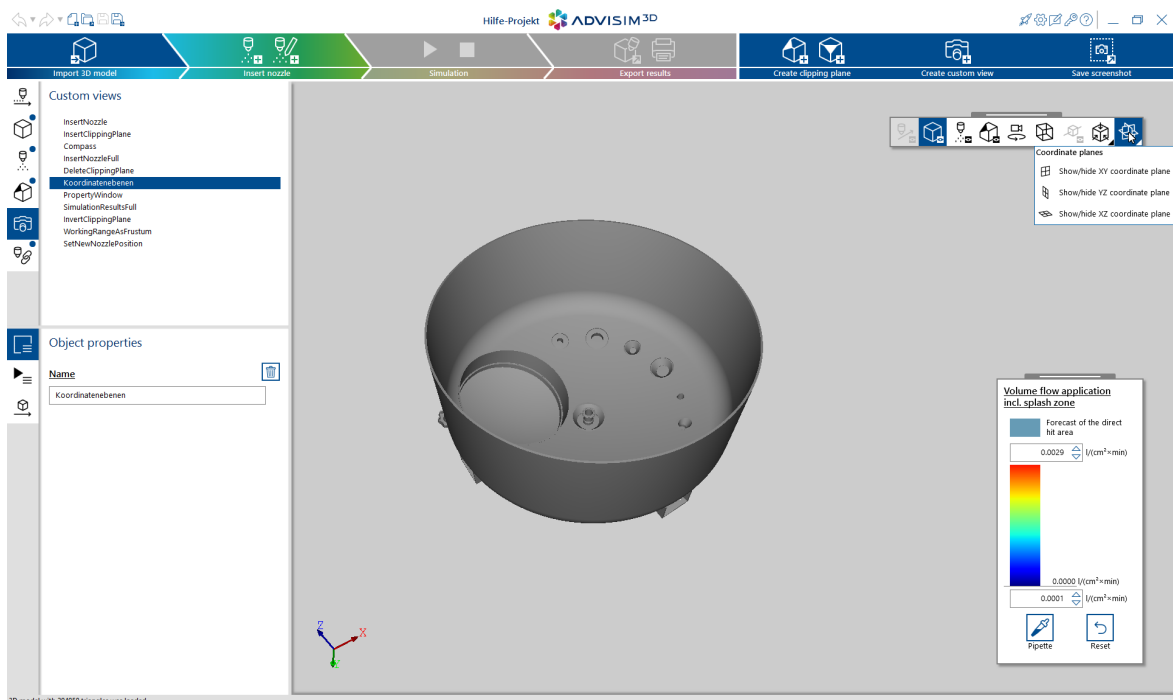
#### 4.8.4 Coordinate system planes

ADVISIM<sup>3D</sup> offers the possibility to display the 3 coordinate system planes.

[BILD EINFÜGEN]

The thicker lines are the axes of the coordinate system.

You can switch all 3 layers (XY, XZ, YZ) on and off independently.



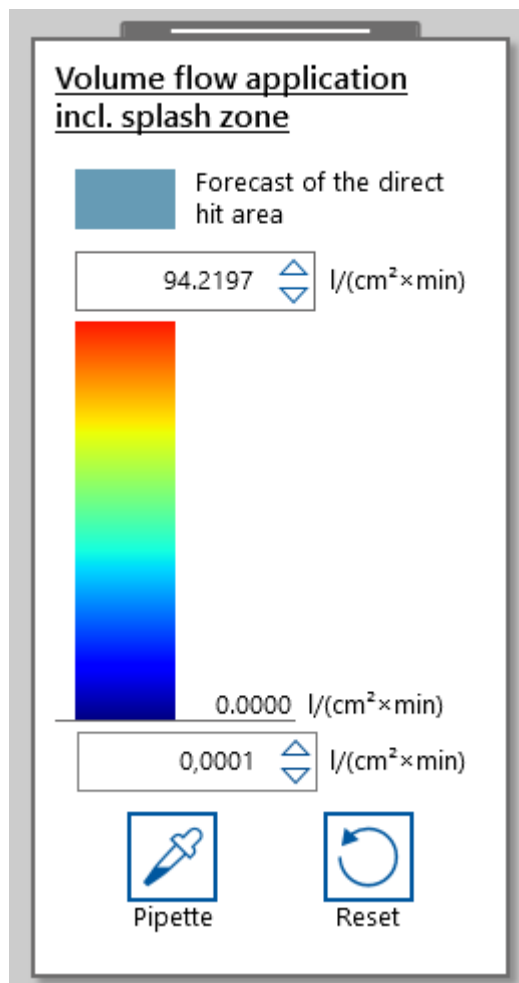
## 4.9 Color scale area

The color scale shows in which range the values of the simulation are located.


Here you can change the maximum and minimum to adapt the displayed simulation result to your preferences.

Incorrect entries have no effect to the displayed result view and are indicated by a validation error "⚠️". If you move with the mouse over the error symbol "⚠️", you will receive a detailed description for the incorrect validation.

**Note:** If you decrease the maximum, all values above will be reduced to the maximum. If you increase the minimum, all values below will be cut off. The simulation thus treats these areas as if they had not been hit.



If you change the color scale, this will affect all results views. Minimum and maximum are only changed for the current results view.

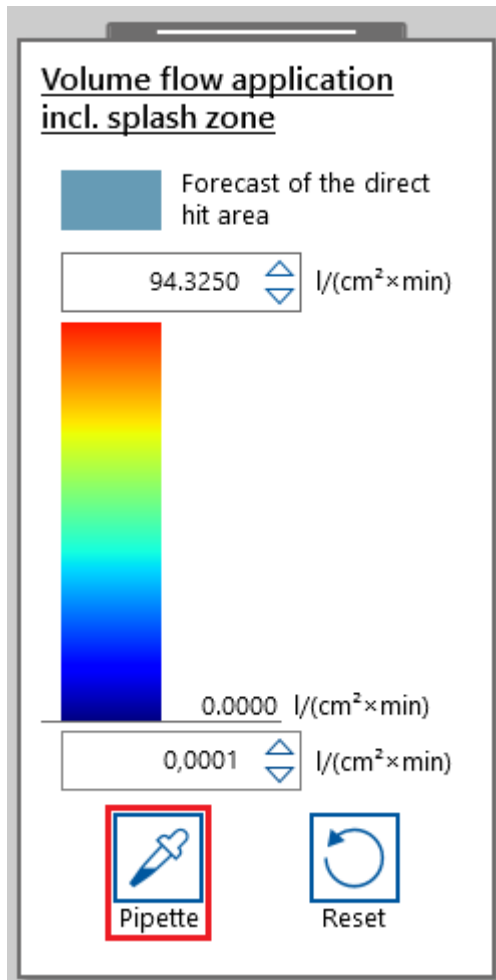
Clicking the -button resets the input fields to the values given by the simulation. Minimum and maximum are saved in the project file.

**Note:** When leaving the simulation, the minimum and maximum values are discarded.

### 4.9.1 Color pipette

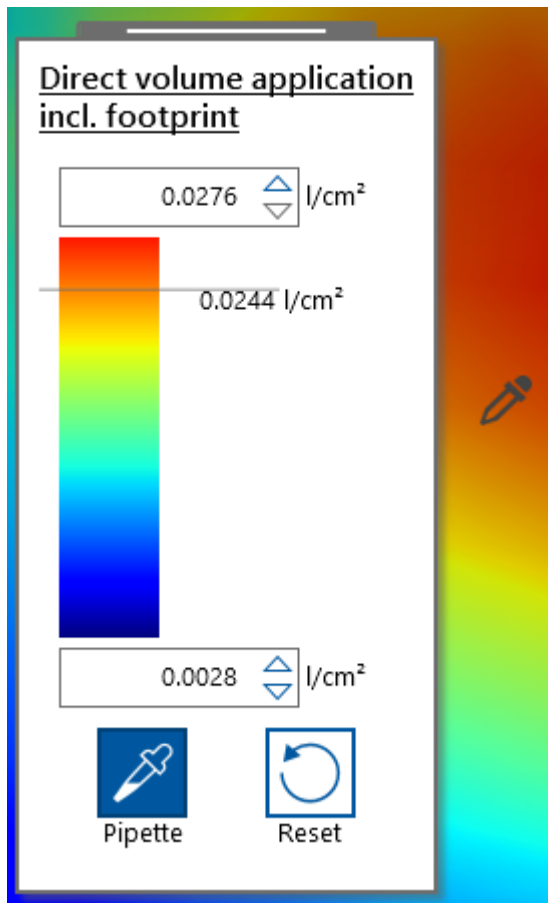
The color pipette shows you the current value under the mouse. This allows you to determine the exact values. The pipette mode only determines values after a nozzle has been placed.

Activate the pipette by activating the pipette button in the color scale area. To deactivate the mode, click the pipette button again.



The tool is active immediately. You only have to move the mouse to the desired position in the 3D view.

Furthermore, the color scale area displays the current simulation value as a numerical value on the color scale.



## 4.10 Model part interaction

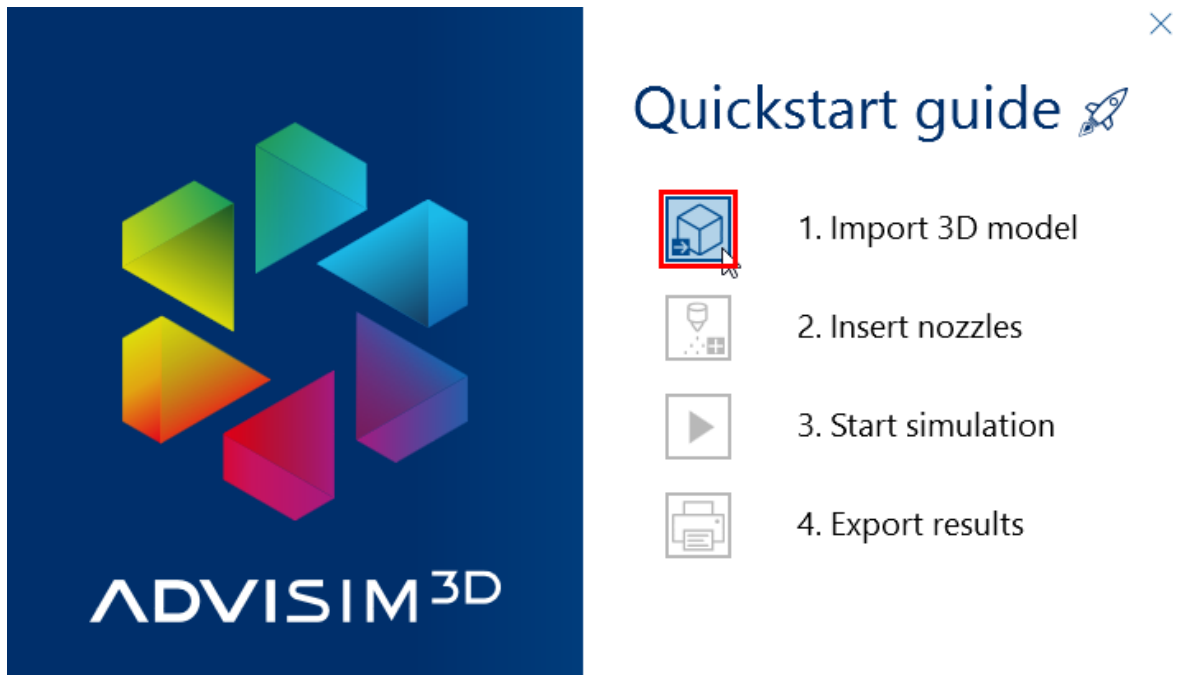
In ADVISIM<sup>3D</sup> it is possible to interact with the model parts [of the loaded STEP file](#). In this way it is possible to [hide individual components](#).

Note that the hidden components are not considered for the [simulation](#).

If you have already added nozzles as components in your 3D model, ADVISIM<sup>3D</sup> also offers you the possibility to [replace them with nozzles](#).

### 4.10.1 Model import

ADVISIM<sup>3D</sup> offers the possibility to import models in STEP file format. To do so, click the button Import to 3D model in the Quick Start Guide (marked red in the following image).



Alternatively, you can click on the “Import 3D model” button in the workflow area:

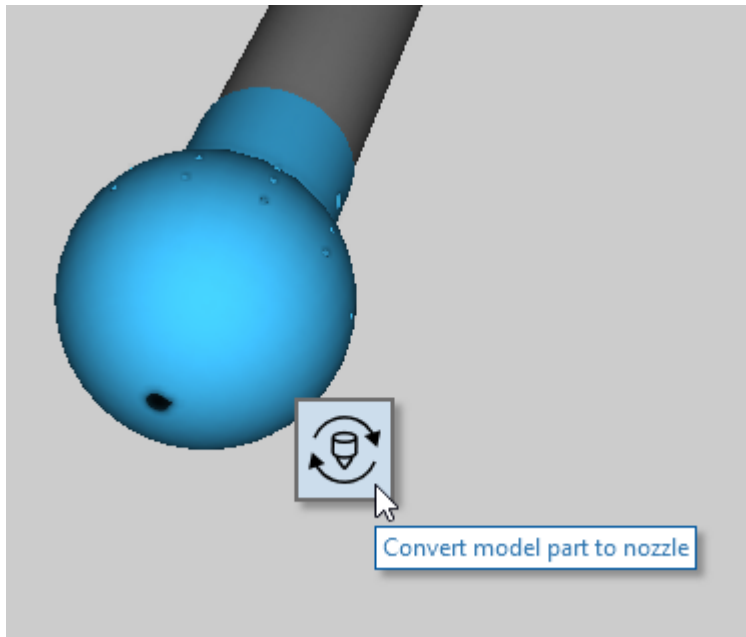


No matter which way you start the import, a file selection dialog appears where you can open files with the extension .step and .stp. Select your desired file and confirm to import it into ADVISIM<sup>3D</sup>.

#### 4.10.2 Convert model part to nozzle

Especially for STEP files that already contain nozzle representations, it makes sense to simply convert them to the desired nozzle type.

To do this, move the mouse over the model part to be replaced and right-click to open the context menu.



If you click the “Convert model part to a nozzle” button, the [nozzle selection dialog](#) opens.

Here you can select the nozzle type to be displayed instead of the model part.

The converted model parts are saved in the project file.

Please note that if you convert a model part into a nozzle and the [linear movement](#) with a distance to origin is active, the nozzle will be placed with the same distance in the scene. The advantage of this is that if you deactivate the linear movement, the nozzle will be at the expected starting point.

## 4.11 Nozzle interaction

Nozzles are the central elements in ADVISIM<sup>3D</sup> to determine the cleaning success.

The first step when working with nozzles is to [insert the nozzle](#) after you have loaded a [STEP file](#). Then you can interact with the nozzle in the scene: For example, you can [select](#) and [move](#) them.

### 4.11.1 Nozzle category

ADVISIM<sup>3D</sup> distinguishes between different nozzle categories, whose different properties are also named.

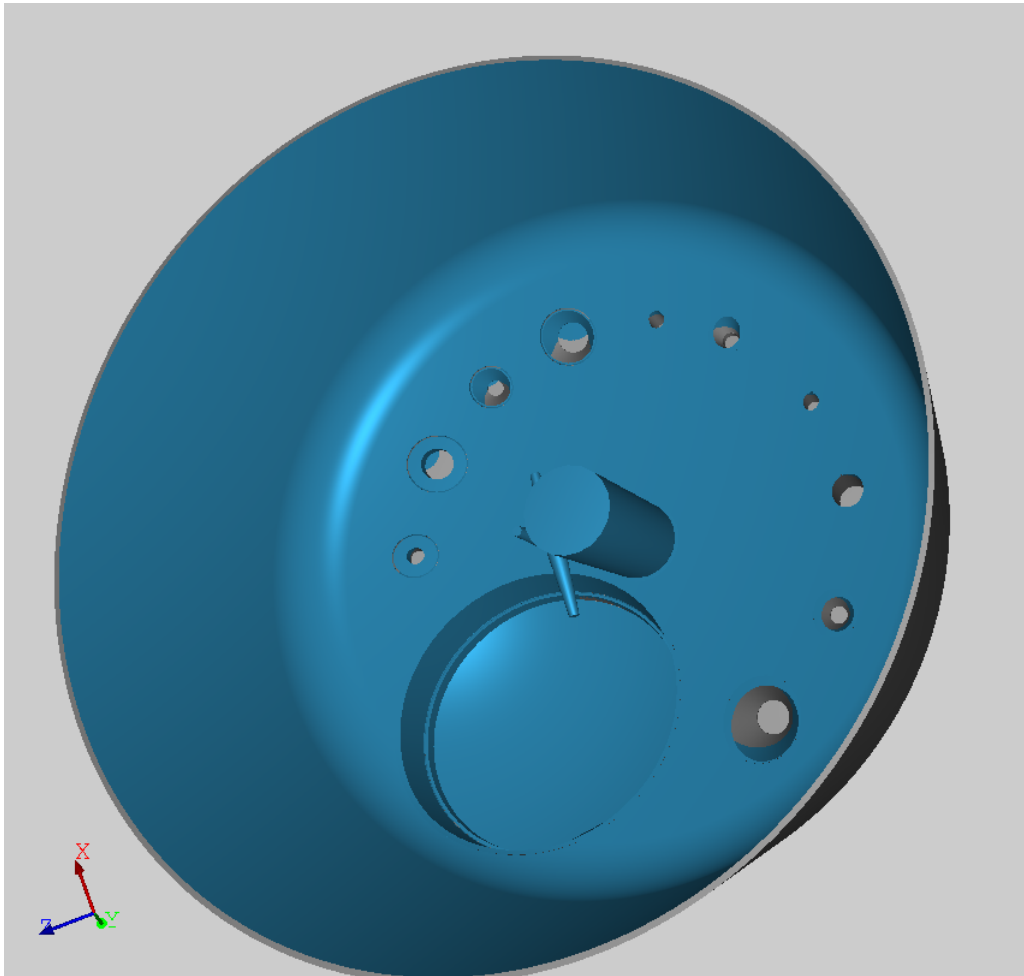
#### 4.11.1.1 Target stream cleaner

In ADVISIM<sup>3D</sup> nozzles are represented with their connector.

As shown in the picture, the nozzle is inserted in the direction of the click point. The Nozzle rotates around herself and around the connector. This connection piece will also create a spray shadow.

This enables you to create more realistic scenarios in ADVISIM<sup>3D</sup>.

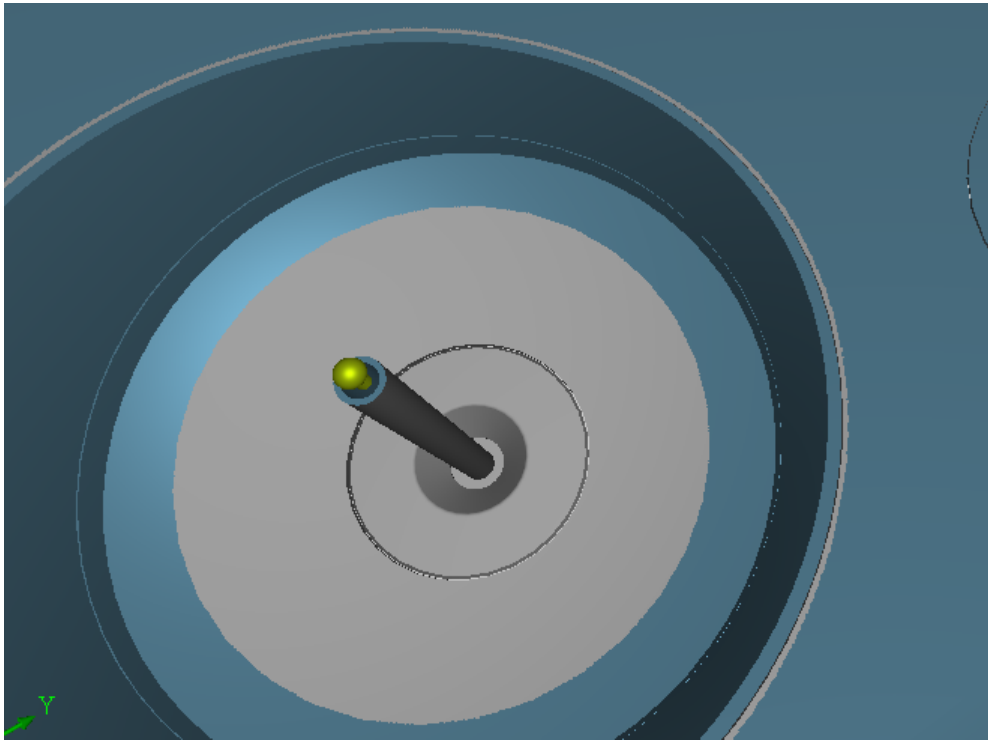
To facilitate positioning, a spray pattern is displayed that would correspond to a nozzle cleaning cycle. This means that direct spray shadows are shown immediately.



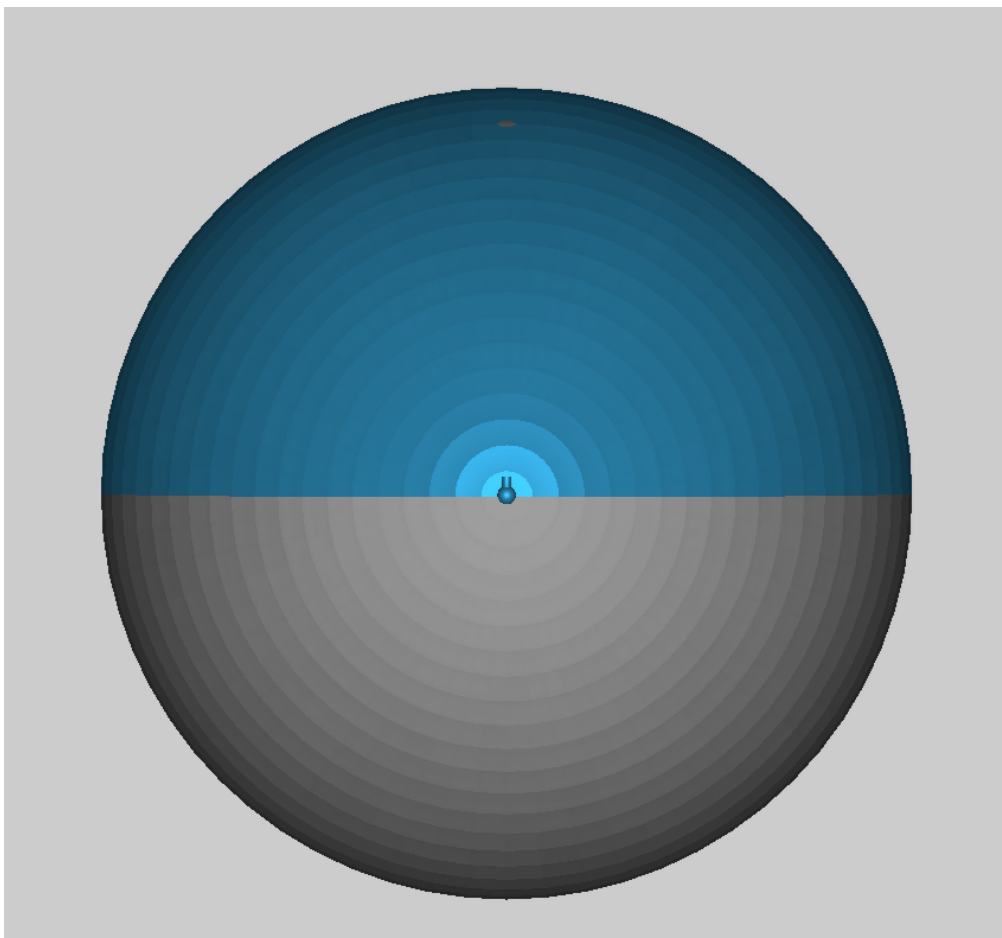
#### 4.11.1.2 Spray balls

For the spray ball, a simplified 360° coverage is displayed in the preview.

In this way an accurate estimation of the positioning is possible without having to perform a simulation.



ADVISIM<sup>3D</sup> also supports sprayballs with various spray angles to the top or bottom, like 180° to top as shown below.





### 4.11.2 Insert nozzle

To insert a nozzle into the 3D scene, select the nozzle insert button in the workflow area.



Alternatively, you can also use the corresponding button in the [Quick Start Guide](#). This opens the [selection dialog](#) for the nozzles.

## Nozzle insertion mode

Selecting a nozzle starts the nozzle insertion mode and the mouse cursor switches to a magic wand



. You use the wand to select a reference point on the model in the 3D scene that will be used for nozzle placement and orientation.

The nozzle is placed centrally between the current view and this reference point, provided this distance is within the workflow area of the respective nozzle.

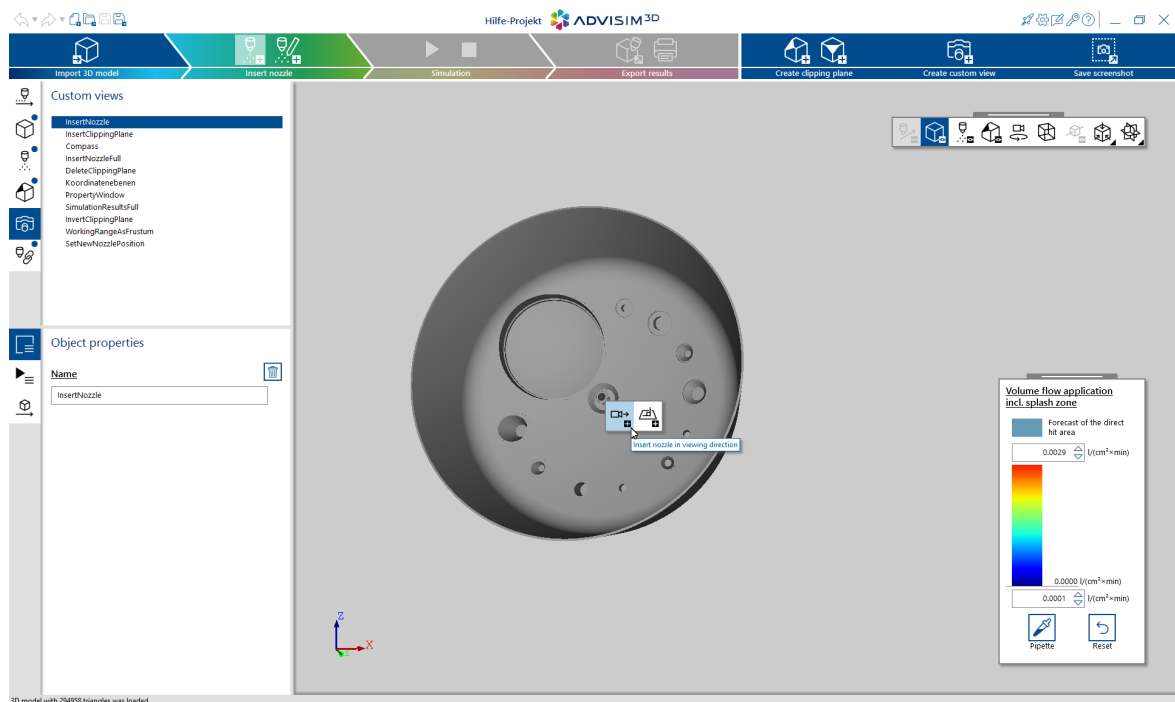
If this is not the case, the nozzle is placed in the middle of its working area.

The nozzle opening is aligned with the reference point. For spray balls and rotation nozzles, the reference point is inverted and refers to the nozzle connection.

To exit the nozzle insertion mode, click on the button again, press the **ESC** key, or click in an empty area of the scene.

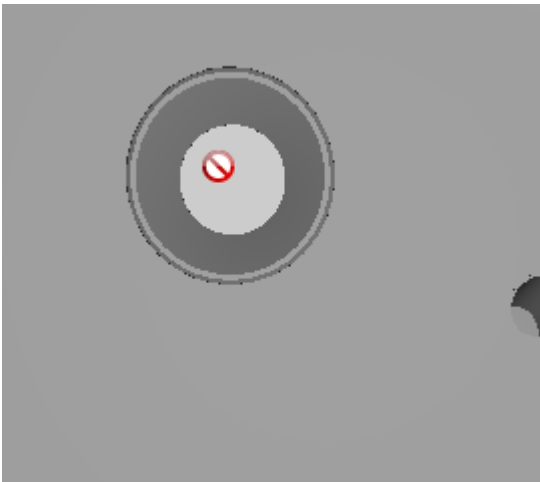
Furthermore you have the possibility to open the context menu with a right click to get more options.

This allows you to insert the nozzle perpendicular to the clicked point. This is also possible if you hold down the **CTRL** key while pasting. You also have the possibility to place the nozzle in the viewing direction.



In addition to the display in the 3D scene, the nozzle is now also shown in the structure tree.

If the nozzle insertion mode is active, the mouse pointer will indicate whether an object is hit or not.



## Convert model part into nozzle

Another possibility is the [conversion of a model part into a nozzle](#). Here the orientation of the 3D model part is directly transferred to the nozzle.

### 4.11.2.1 Nozzle selection dialog

In the nozzle selection dialog, you can choose from a variety of nozzles and then use them in ADVISIM<sup>3D</sup>.

Click on the desired nozzle in the table and then press the "Apply" button or double-click on the nozzle. The nozzle selection dialog closes and ADVISIM<sup>3D</sup> is now in [insert nozzle mode](#).

Select nozzle type

Filter

Search

Manufacturer

☐ ADVISIM3D Beispieldüsen

☐ AWH

☐ Fraunhofer IVV

☐ Hake

☐ Scanjet Systems

☐ Alfa Laval

☐ Bete

☐ GEA

☐ Kohiko

☐ Schwarte

☐ Aqua Duna

☐ Ficonnex

☐ Guangzhou Cleaningspray Equipment

☐ Lechler

☐ Spraying Systems

Category

☐ Flat spray nozzle

☐ Spray ball

☐ Full cone nozzle

☐ Rotation nozzle

☐ Hollow cone nozzle

☐ Spate cleaner

No search criteria was selected. All previously used nozzles are displayed.

Description	Manufacturer	Item number	Nozzle category	Spraying angle	max. working range	Volume flow	Pressure range	Connector type
Flachstrahldüse 45°	ADVISIM3D Beispieldüsen	flktiv	Flat spray nozzle	45°	500 mm	1 l/min @ 2 bar	2 - 5 bar	
Vollkegeldüse 60°	ADVISIM3D Beispieldüsen	flktiv	Full cone nozzle	60°	725 mm	1 l/min @ 3 bar	2 - 6 bar	
IntenseClean	Lechler	STM.406.1YAS	Rotation nozzle	360°	9000 mm	221 l/min @ 5 bar	2 - 7 bar	
Sprühkugel Baureihe 591	Lechler	591.A21.17	Spray ball	360°	400 mm	181 l/min @ 2 bar	0.5 - 5 bar	

Apply

Cancel

## Filtering

You can filter the nozzle catalog by manufacturer and nozzle category by activating the respective selection fields. For example, you can

- list the product catalog of a manufacturer if you select the manufacturer,
- display an overview of the spray balls available from all manufacturers if you activate the "Spray ball" category,

- filter for atomizer nozzles from one manufacturer if you activate the respective manufacturer and the categories "Flat spray nozzle", "Full cone nozzle" and "Hollow cone nozzle"
- carry out a comparison of Rotation nozzles from two manufacturers if you activate the two manufacturers and the "Rotation nozzle" category.

### [BILD EINFÜGEN]

If no filter option is selected, all nozzle types you have used so far are listed.

## Searching



You can search by criteria such as the manufacturer or article number. Select the "Search" tab and enter the search term in the text field. Limit the search result by separating several search terms with "," (comma without the quotation marks and spaces).

 Filter
  Search

## Sorting


To sort the nozzle catalog by a nozzle property, click on the header of the respective column. Each additional click on this column header reverses the sort order. This is indicated by the arrow next to the column label.

In the example, the catalog was sorted according to the maximum working range in ascending order:

	Description	Manufacturer	Item number	Nozzle category	Spraying angle	max. working range 	Volume flow	Pressure range	Connector type
	Flachstrahldüse 75°	ADVISIM3D Beispieldüsen	fiktiv	Flat spray nozzle	75°	500 mm	1 l/min @ 2 bar	2 - 5 bar	
	Flachstrahldüse 120°	ADVISIM3D Beispieldüsen	fiktiv	Flat spray nozzle	120°	500 mm	1 l/min @ 2 bar	2 - 5 bar	
	Vollkegeldüse 90°	ADVISIM3D Beispieldüsen	fiktiv	Full cone nozzle	90°	650 mm	1 l/min @ 2 bar	0.5 - 4 bar	
	Vollkegeldüse 60°	ADVISIM3D Beispieldüsen	fiktiv	Full cone nozzle	60°	725 mm	1 l/min @ 3 bar	2 - 6 bar	
	Vollkegeldüse 45°	ADVISIM3D Beispieldüsen	fiktiv	Full cone nozzle	45°	750 mm	1 l/min @ 2 bar	0.5 - 5 bar	
	Hohlkegeldüse 60°	ADVISIM3D Beispieldüsen	fiktiv	Hollow cone nozzle	60°	800 mm	1 l/min @ 2 bar	1 - 9 bar	
	Vollkegeldüse 25°	ADVISIM3D Beispieldüsen	fiktiv	Full cone nozzle	25°	800 mm	1 l/min @ 2 bar	1 - 7 bar	
	Sprühkugel 180° nach Unten	ADVISIM3D Beispieldüsen	fiktiv	Spray ball	180°	2500 mm	5 l/min @ 2 bar	1 - 4 bar	
	Sprühkugel 120° nach Unten	ADVISIM3D Beispieldüsen	fiktiv	Spray ball	120°	2500 mm	5 l/min @ 2 bar	1 - 4 bar	
	Sprühkugel 270° nach Unten	ADVISIM3D Beispieldüsen	fiktiv	Spray ball	270°	2500 mm	5 l/min @ 2 bar	1 - 4 bar	
	Sprühkugel 240° nach Unten	ADVISIM3D Beispieldüsen	fiktiv	Spray ball	240°	2500 mm	5 l/min @ 2 bar	1 - 4 bar	
	Sprühkugel 180° nach Oben	ADVISIM3D Beispieldüsen	fiktiv	Spray ball	180°	2500 mm	5 l/min @ 2 bar	1 - 4 bar	
	Sprühkugel 120° nach Oben	ADVISIM3D Beispieldüsen	fiktiv	Spray ball	120°	2500 mm	5 l/min @ 2 bar	1 - 4 bar	
	Sprühkugel 270° nach Oben	ADVISIM3D Beispieldüsen	fiktiv	Spray ball	270°	2500 mm	5 l/min @ 2 bar	1 - 4 bar	
	Sprühkugel 240° nach Oben	ADVISIM3D Beispieldüsen	fiktiv	Spray ball	240°	2500 mm	5 l/min @ 2 bar	1 - 4 bar	

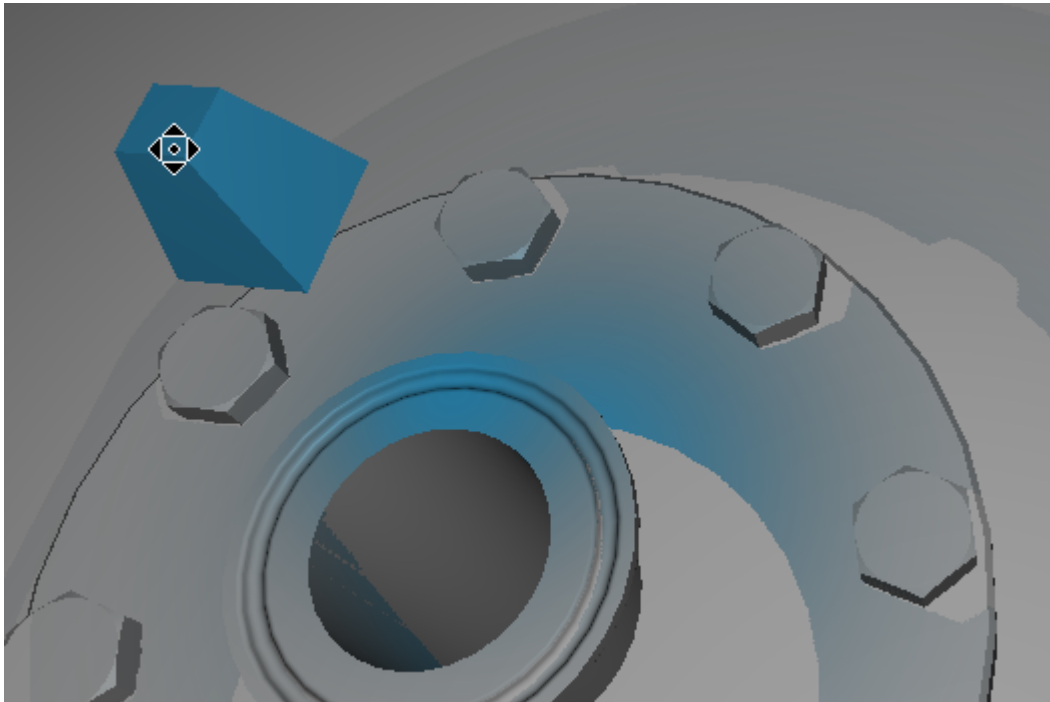
### 4.11.3 Move nozzle

To move a nozzle, it must first be [selected](#) via the structure tree or the 3D view.

If a nozzle can be moved, the mouse pointer changes to .

Hold down the left mouse button to move the nozzle to its new position. To cancel this process, press the **ESC** key while moving.

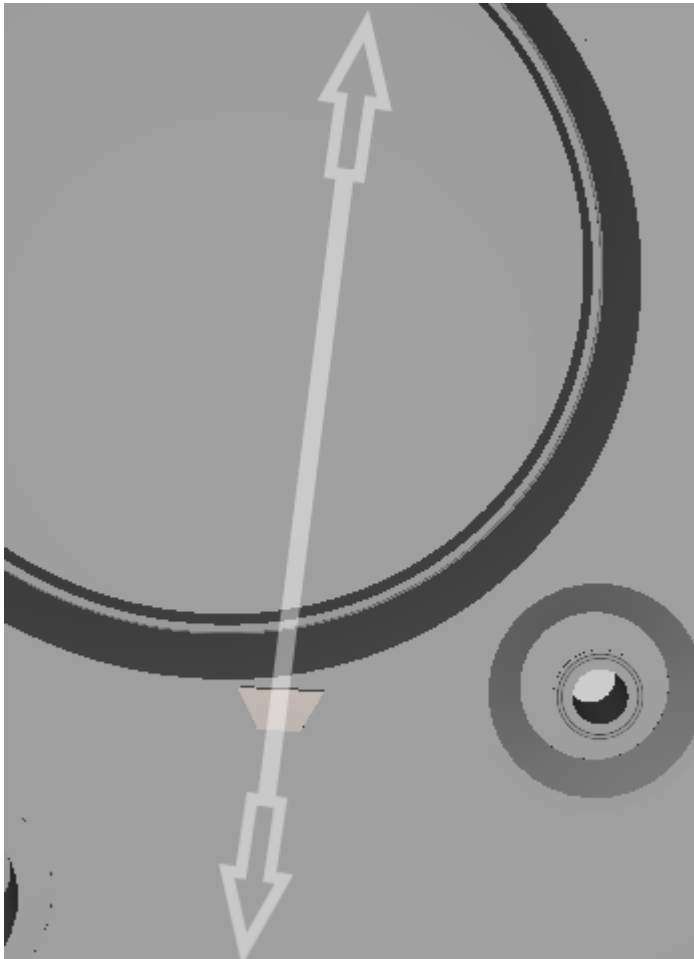
If you release the mouse, the new position of the nozzle is applied



#### 4.11.3.1 Move nozzle along its spray direction

To move a nozzle, it must first be selected.


It can then be moved along its spray direction by holding down the **CTRL** key and simultaneously scrolling the mouse wheel.



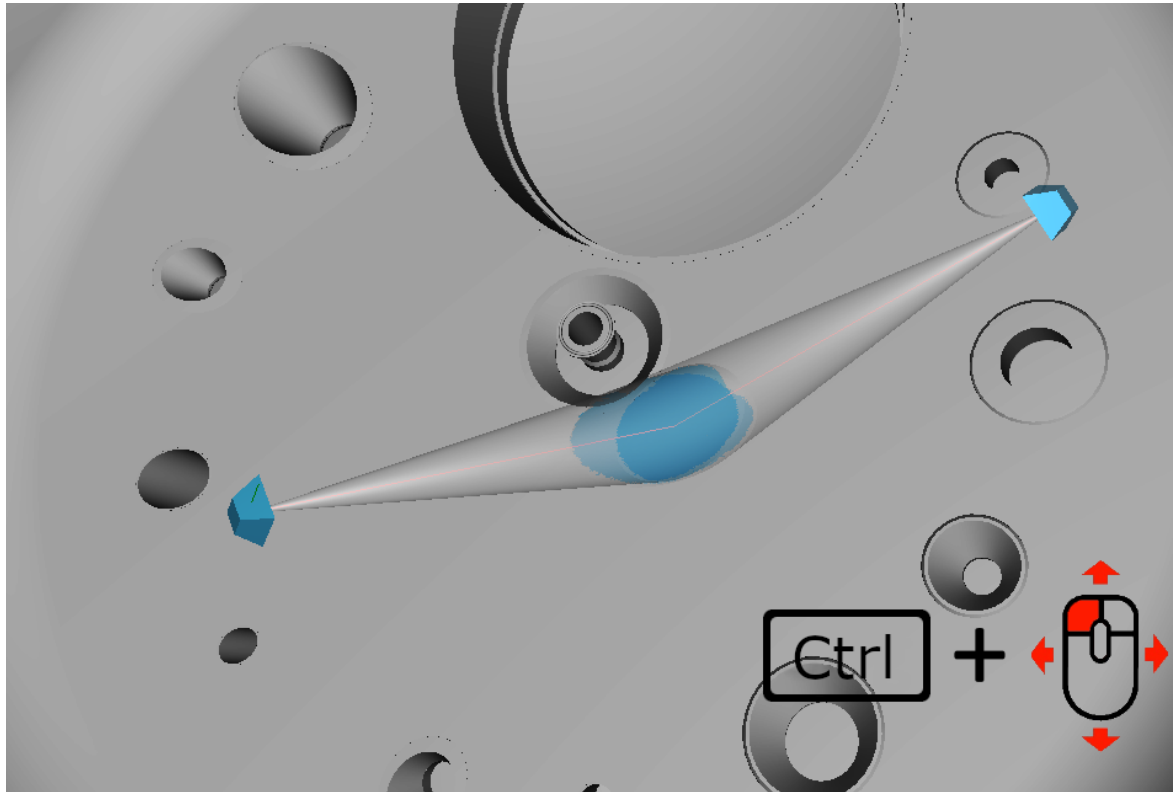
#### 4.11.3.2 Move nozzle with constant target point

A nozzle can also be moved so that the target point of the nozzle does not change.

To do this, select a nozzle and move the nozzle while holding down the **CTRL** key.

Similar to [moving a nozzle](#), the cursor changes to .

Moving the nozzle with a constant target point otherwise behaves the same as normal moving.



Via the context menu of the nozzle it is possible to define the movement behavior of the nozzle. Select the nozzle and open the context menu with the right mouse button:



The -button indicates that the nozzle is moved with constant target point.



The -button indicates that the nozzle is being moved with a constant target point. This behavior is enabled by default.

#### 4.11.4 Change nozzle direction


Geben Sie hier den Text ein.

##### 4.11.4.1 Reset nozzle direction

To reset the spray direction of a nozzle, open the context menu in the 3D scene with the right mouse button.





Select the -button to enter the mode for setting the new nozzle direction.

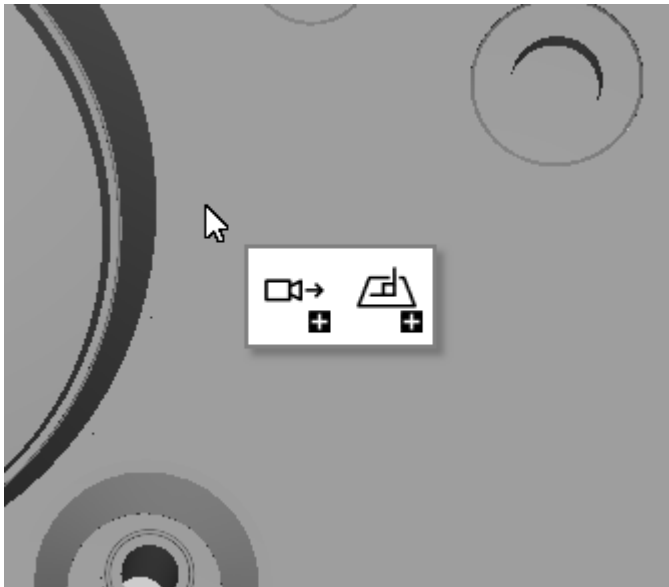
Then you can select a new reference point, like when [inserting a nozzle](#). When the mode is active, the cursor changes to a wand .

While holding down the **CTRL** key, the nozzle is aligned along the surface normal of the clicked plane.

If you have activated the mode, you can open the context menu with a right click. This shows you the two variants with which a nozzle can be realigned:


With  you can select a new destination.

With  you can align the nozzle along the surface normal of the selected target point.



#### 4.11.4.2 Invert nozzle spray direction

To invert the spray direction of a nozzle, open the [context menu](#) in the 3D scene with the right mouse button.

A click on the  -button inverts the direction of the selected nozzle.



#### 4.11.4.3 Turn nozzle around its spray direction

To rotate a nozzle around its own axis, it must first be selected.

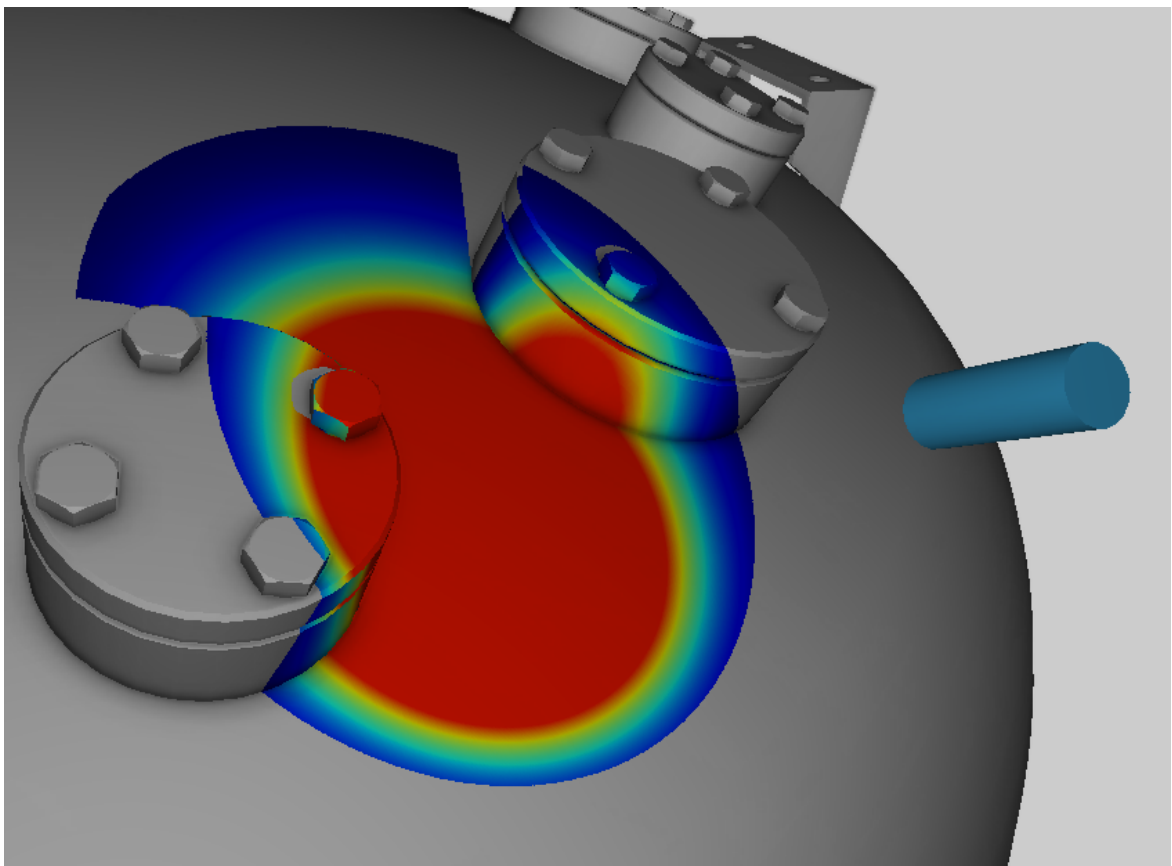
You can now turn this nozzle clockwise or counterclockwise in the [properties area under up-vector](#).

#### 4.11.5 Select nozzle

ADVISIM<sup>3D</sup> offers two possibilities to select a nozzle:

- Selection of the nozzle via the structure tree
- Selection by left-clicking or right-clicking on the nozzle in the 3D scene



A selected nozzle is highlighted in blue and its spray pattern is portrayed over the overall result.



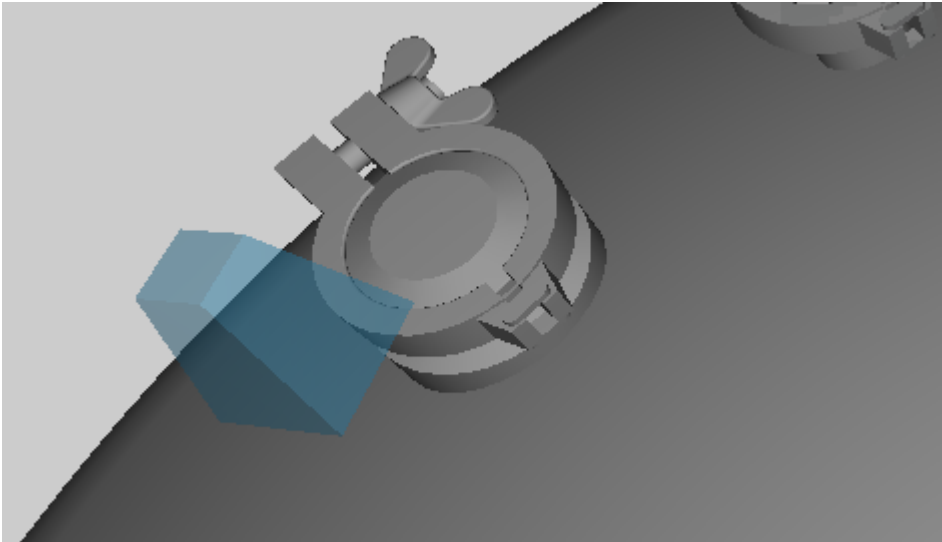


### 4.11.6 Show and hide nozzle

Nozzles can be shown or hidden. If a nozzle is hidden, its nozzle model and spray pattern are no longer displayed. Furthermore, it is not considered for the simulation. It is then displayed as hidden in the structure tree. The visibility of a nozzle is saved in the project and restored upon loading.

To show or hide a nozzle, select it in the structure tree or in the 3D scene and click the  or  -button of the desired nozzle in the nozzle tab of the structure tree.


A selected and hidden nozzle is displayed in transparent blue.



The button for switching the visibility status of all nozzles at the same time is located in the 3D toolbar:



### 4.11.7 Change nozzle type

To change the type of a nozzle, select the nozzle and click the -button (as shown below) located in the [properties area](#). Alternatively, the same button is located in the context menu of the nozzle, which you can open by right-clicking on the nozzle.



Alternatively, click the delete key to delete a [selected nozzle](#) or use the button in the properties area.

### 4.11.9 Export nozzle

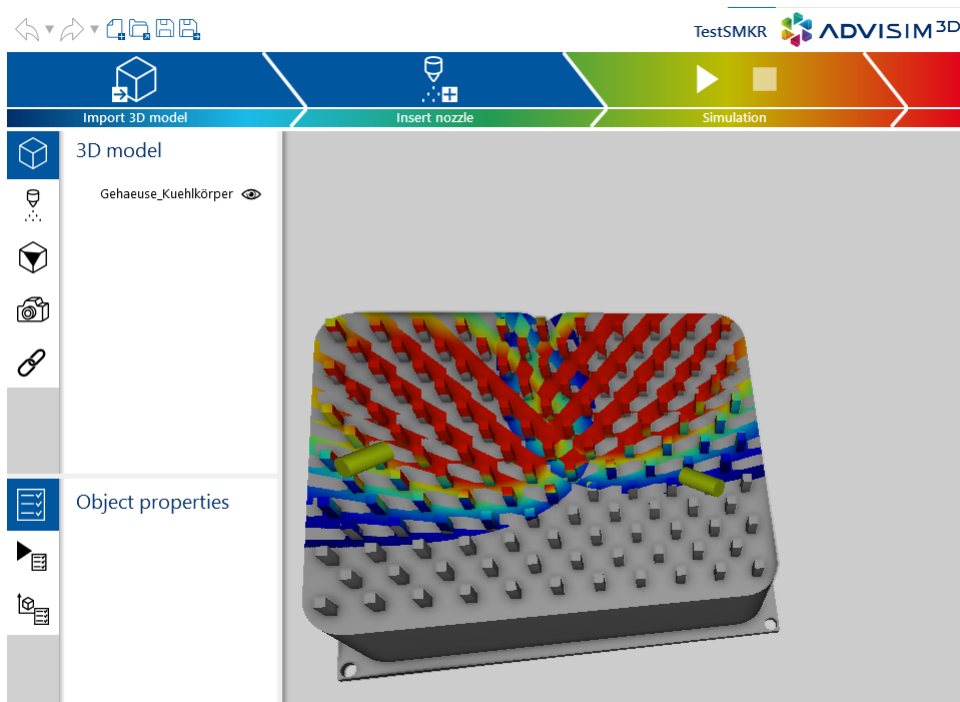


To export nozzles, select the  - button in the workflow area. After you have selected a storage location, the nozzles are exported in STEP format.

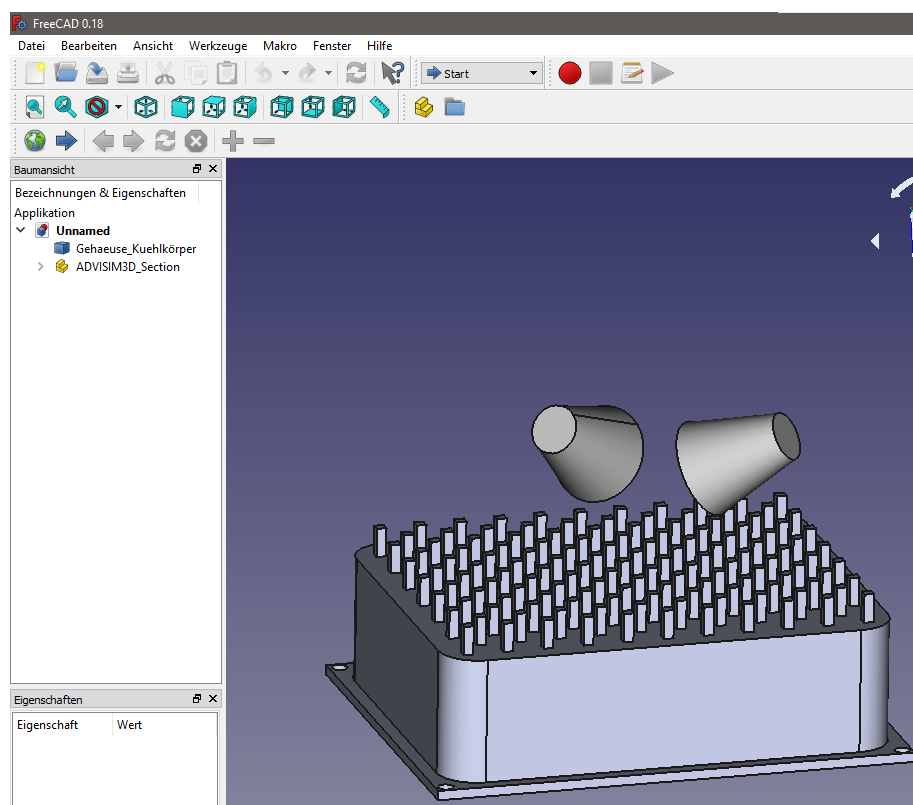
The button for exporting nozzles is only active if nozzles are in the scene and are not hidden. Only visible nozzles can be exported.

Exported nozzles can be used in other CAD programs.

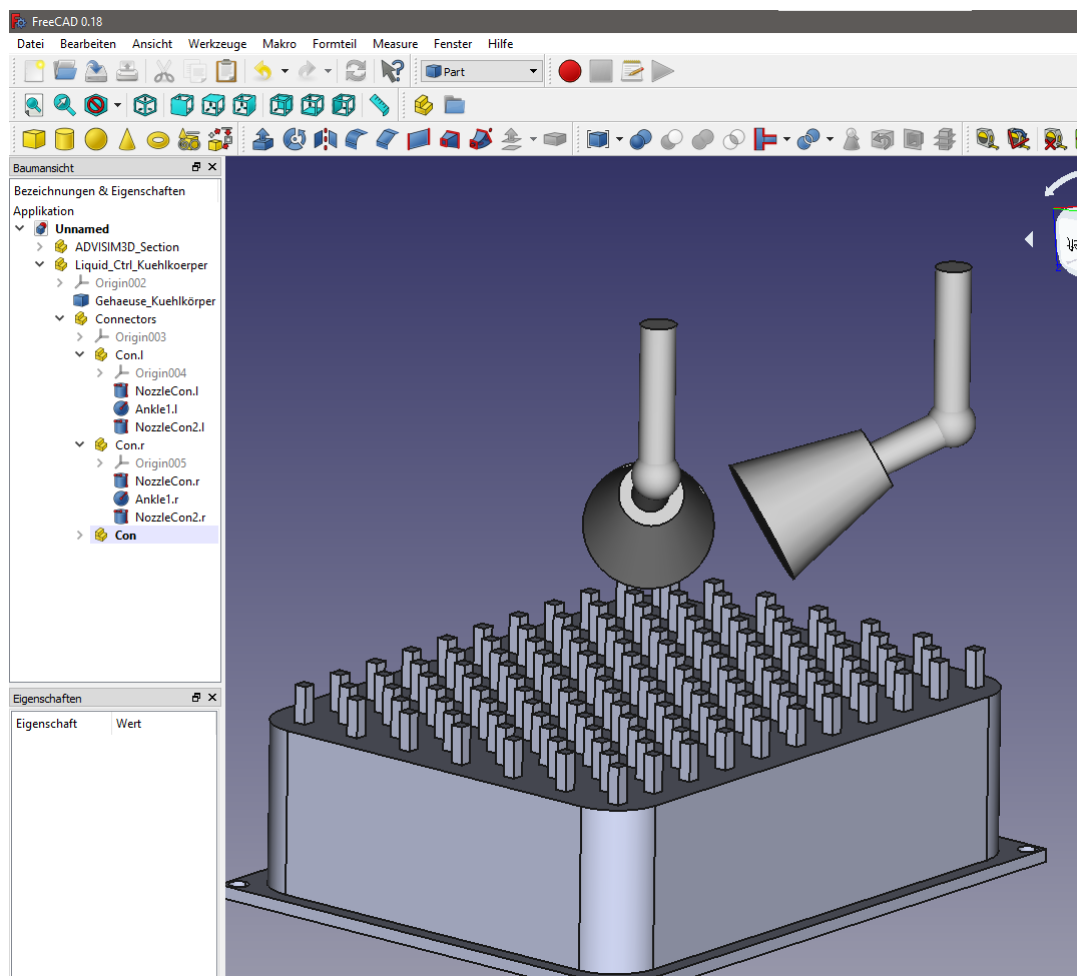
- Save your project in ADVISIM<sup>3D</sup>



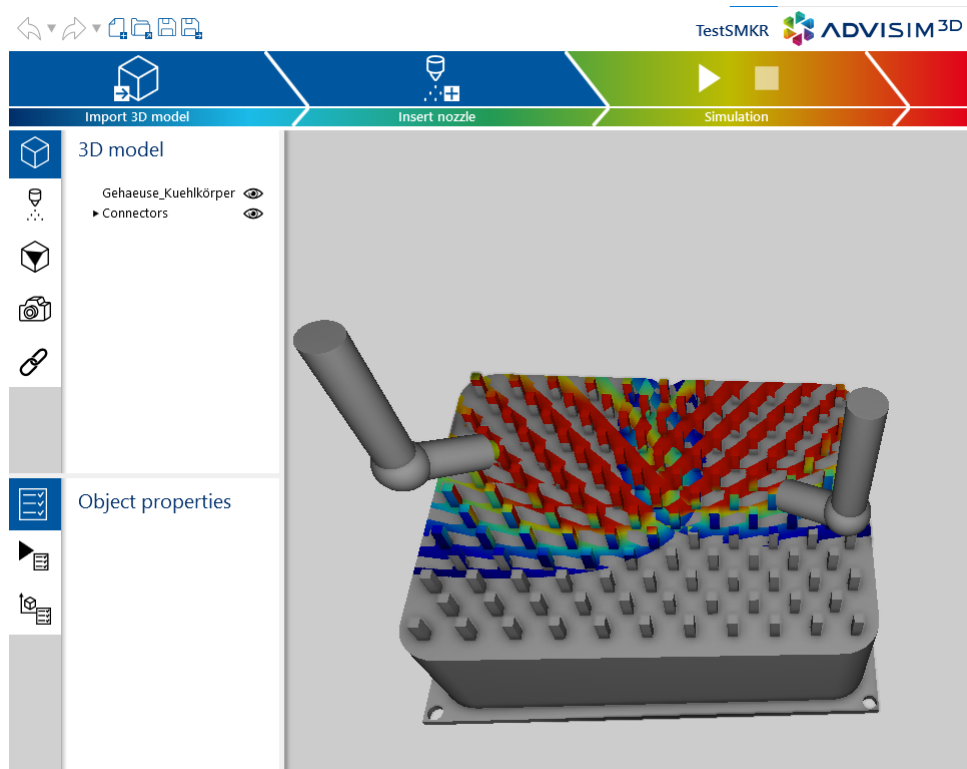
- Open the 3D-Model in your CAD program
  - Import the exported nozzles into your CAD program
  - The alignment and position of the nozzles is taken from ADVISIM<sup>3D</sup>, so that you can make changes to the model



- Export the modified model




- Go back to ADVISIM<sup>3D</sup>
  - If you closed ADVISIM<sup>3D</sup> reopen the program and load the saved project
- Import the modified model

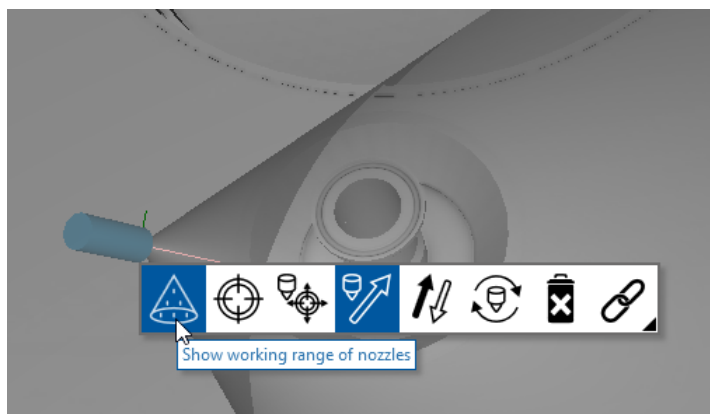


When re-importing the model with nozzles, these are automatically recognized and filtered out as ADVISIM<sup>3D</sup> nozzles.

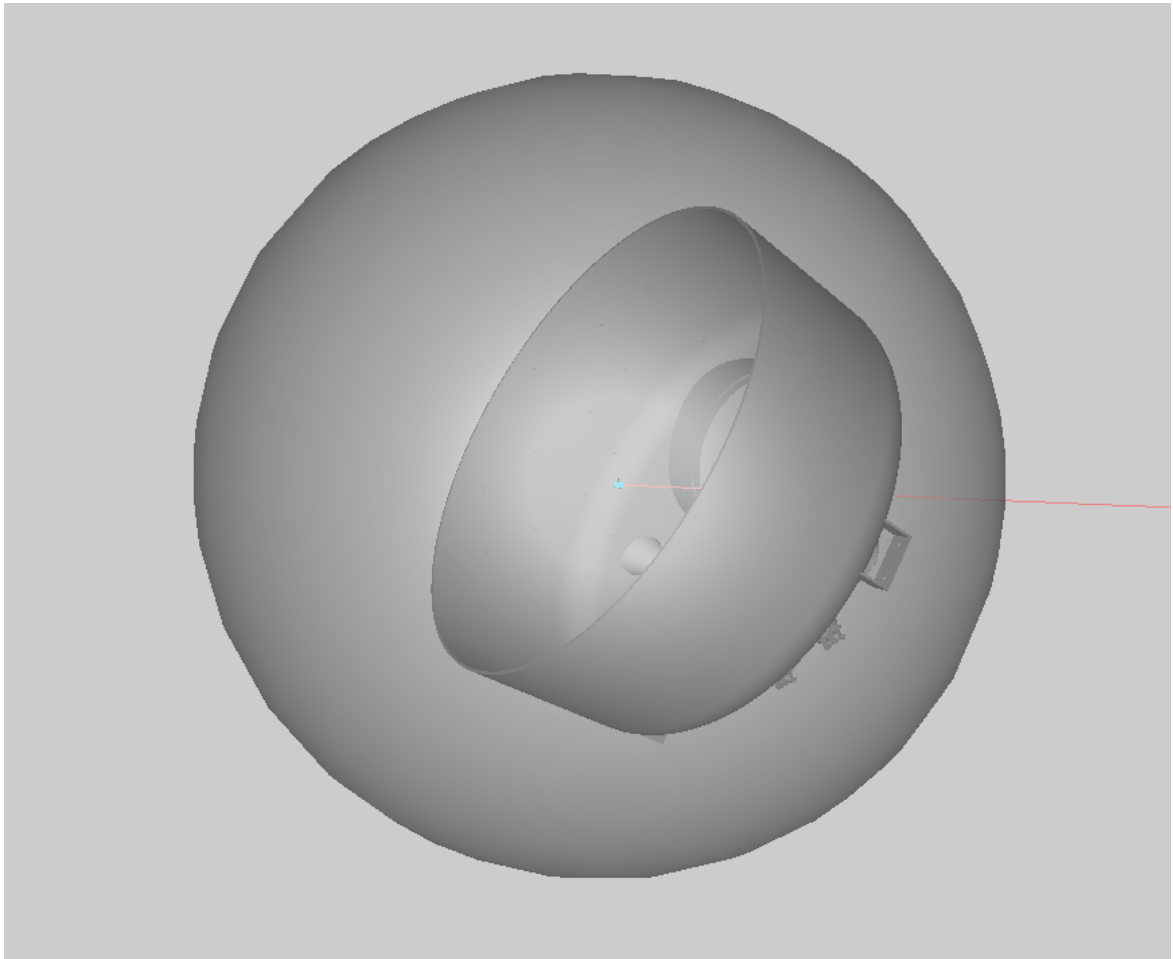
**Caution!** A change to the ADVISIM<sup>3D</sup> assembly (Advisim\_Section > Advisim\_Nozzles) or the nozzle designation (Nozzle\_[#ID]) can result in the nozzles no longer being recognized during a re-import to ADVISIM<sup>3D</sup>.

#### 4.11.10 Show working range

To display the working range of the selected nozzle, click the -button in the [context menu of the nozzle](#).



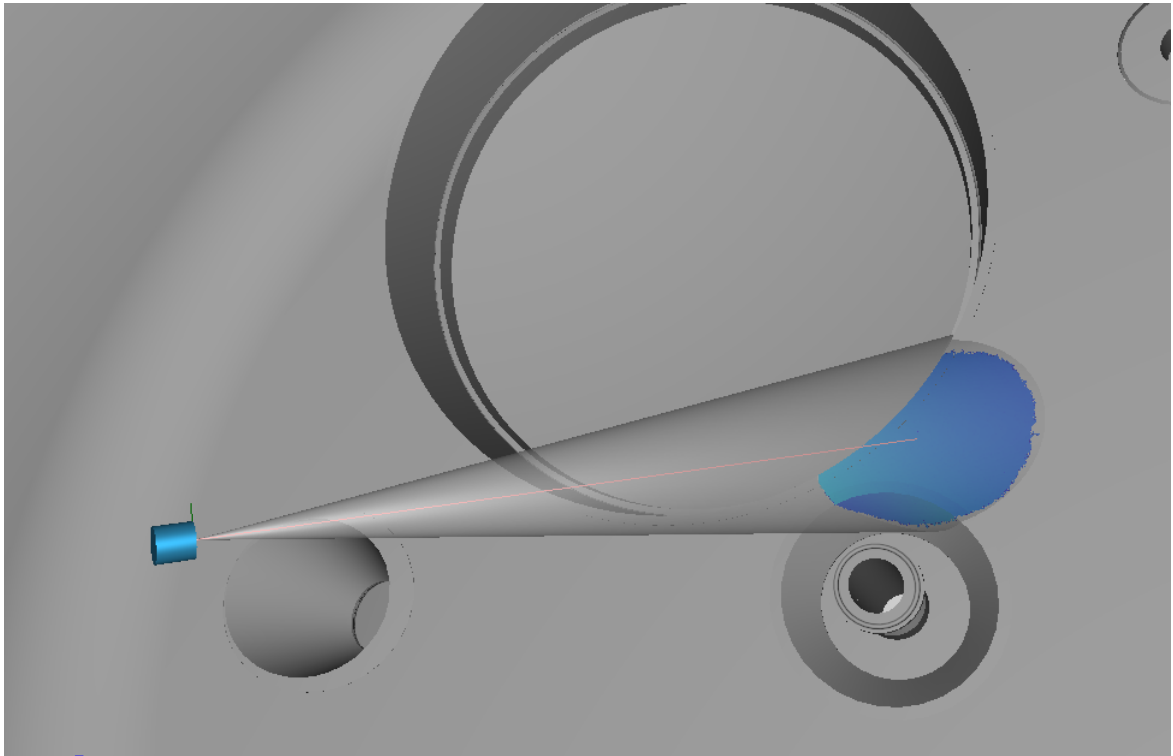
For atomizer nozzles, the working range of a nozzle is displayed as a cone. For spray balls and target stream cleaners as a sphere.



The semi-transparent gray cone indicates the effective range of the respective nozzle.

The red stream is the focus stream of the nozzle. It is also used for distance measurement between the selected nozzle and the 3D-Model.

The up-vector of the nozzle is shown as a green line on the nozzle and serves for orientation in space.

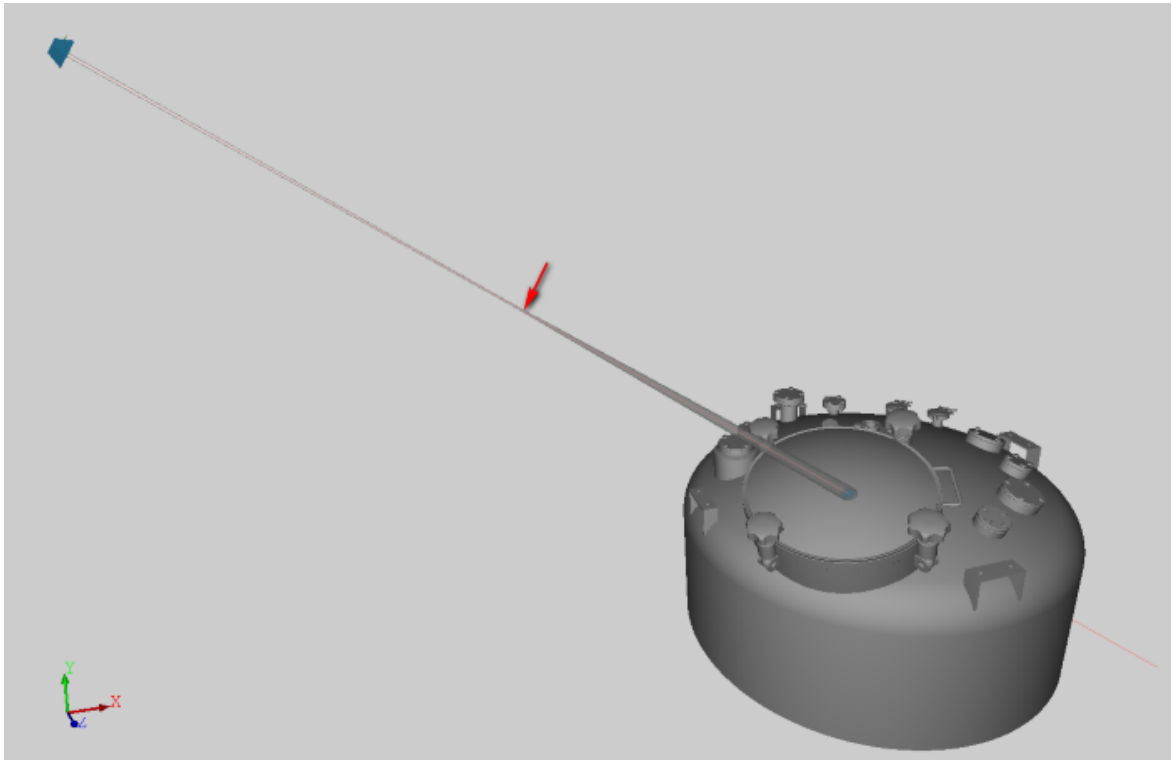


## Solid stream

The working area of solid stream nozzles is displayed specifically to distinguish the area of the non-decaying solid stream from that of the decaying round stream.

The red arrow marks the transition from the non-decaying solid stream to the decayed solid stream. ADVISIM<sup>3D</sup> models and calculates a continuous linear transition from the non-decaying to the decaying solid stream.





#### 4.11.11 Customize and create spray balls

ADVISM<sup>3D</sup> offers you the option of creating spray balls according to your own requirements. Select an existing spray ball from the [nozzle catalog](#) or design a completely new spray ball from scratch. Individual spray balls can be changed or deleted at any time.

The spray balls you create fit seamlessly into the familiar ADVISM<sup>3D</sup> operating concept, i.e. they appear in the [nozzle selection dialog](#) and are ready for you to use in the 3D scene.

Individual spray balls are linked to your user license and are available to all users of your license via the [nozzle catalog](#). They can be viewed, used, changed or deleted by them.

To help you get started quickly with creating individual spray spheres, you will find a quick start guide under [First steps](#).

##### 4.11.11.1 First steps

This section shows you how to create a new spray ball in ADVISM<sup>3D</sup>.

### 1. Import STEP file

To activate the function for creating a custom spray ball, [load a 3D model](#) from a STEP file.

### 2. Start dialog

Then click on the "Create and insert nozzle" button in the [workflow area](#):



This opens the [dialog window](#) for creating spray balls. From the [nozzle type template tab](#), select the "New spray ball from scratch" option by clicking on the corresponding button.

### 3. Enter general nozzle parameters

You will see an empty screen with [general nozzle parameters](#). Enter the data in the corresponding fields.

All fields marked with an exclamation mark are mandatory and must contain no errors. You will receive an explanation of input errors if you point to the exclamation mark with the mouse:

#### Pressure

Maximum:

35.0 bar

Standard:

Minimum:

40 bar

#### Volume flow

At maximum pressure:

The value must be greater than the standard pressure.  
The value must be greater than the minimum pressure.

### 4. Enter spray ball parameters

Click on the "[Spray ball parameters](#)" tab. Enter the typical data for your spray ball in this screen. You will also receive an explanation of input errors via the exclamation marks.

Configure the hole rings of the spray ball in the table at the bottom of the screen. Click on the "+" button to add new entries to the table:

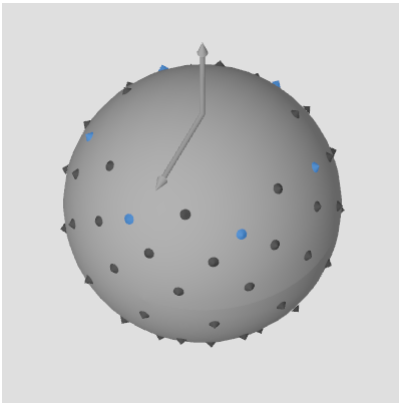


To edit the data of an entry, click on the respective table row and enter the corresponding values in the text fields that appear:

#### Hole rings

+	Latitude	Offset	Number of holes
-	45°	15°	6
-	55	-15	6
-	65°	0°	12

The spray ball preview in the top right-hand area of the screen shows you a live preview of these hole ring configurations.



## 5. Insert nozzle in 3D scene

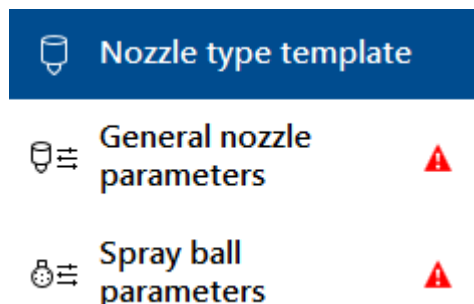
If your entries in the masks are correct, the "Apply" button is activated. Once the entries have been accepted, the spray ball is created and is available in the 3D scene.

## Congratulations!

You have created your first spray ball. Start a [simulation](#) and evaluate the result.

### 4.11.11.2 Input masks

The dialog window for creating and modifying spray balls contains the following three tabs on the left-hand side (click on an entry to obtain detailed information on the respective tab):



To the right of the tab text, exclamation marks indicate that errors have occurred in the data entered in the respective screen. Correct these and the exclamation mark will disappear.

The buttons to accept and cancel the dialog are located at the bottom of the window:



The "Accept" button is only active if none of the tabs contain an error. Press this button to accept the data entered and create the spray ball. The application is then in [nozzle insertion mode](#) so that you can immediately place the newly created spray ball in the 3D scene.

The "Cancel" button cancels the dialog, the data entered is discarded and no custom spray ball is created.

## 4.11.11.2.1 Nozzle type template

In this screen, you select whether you want to create a custom spray ball from scratch or based on an existing spray ball in the ADVISIM<sup>3D</sup> nozzle catalog.

**Nozzle type template**

New spray ball from scratch

New spray ball based on existing one

## Create spray ball from scratch

Select the "New spray ball from scratch" option. The "[General nozzle parameters](#)" and "[Spray ball parameters](#)" tabs appear. The exclamation marks next to the tab text inform you that the text fields must be filled in with the spray ball data. The dialog can then be accepted and the spray ball is added to the [nozzle catalog](#).

## Creating spray ball based on an existing spray ball

Select the option "New spray ball based on an existing one". The [nozzle selection dialog](#) opens and all spray balls available in the nozzle catalog are displayed. Click on the desired spray ball and press the "Apply" button. The "[General nozzle parameters](#)" and "[Spray ball parameters](#)" tabs appear in the dialog window for creating the spray ball. The respective text fields are pre-assigned with the data of the selected nozzle. Change individual values and accept the dialog. A copy of the selected spray ball is added to the [nozzle catalog](#) as a new spray ball.

## 4.11.11.2.2 General nozzle parameters

In this screen, you can enter the general parameters of custom spray balls. The screen is divided into the sections described below.

Catalog data

Description:	Spray ball 360°
Manufacturer:	ADVITEC
Item number:	AIG-SK-591
Connector type:	EN 10226 R 1/8

Geometry

Height:	53.0	mm
Diameter:	40.0	mm

Working range

Maximum:	400	mm
Minimum:	1	mm

Pressure

Maximum:	5.0	bar
Standard:	2.0	bar
Minimum:	0.5	bar

Volume flow

At maximum pressure:	286.19	l/min
At standard pressure:	181.00	l/min

Catalog data

This section contains the catalog data for the spray ball.

**Description:** the name of the spray ball  
The text can be freely selected, but must not be empty.

**Manufacturer:** the manufacturer of the spray ball  
The text can be freely selected, but must not be empty.  
If you create your own spray ball [from scratch](#), the name stored in your license is entered here. If you are using an [existing spray ball](#), its manufacturer will be used. In both cases, you can change the entry as you wish. Please note, however, that the manufacturers are listed as a filter option in the [nozzle selection dialog](#). Too many different manufacturer entries can make the filter option confusing.

**Article number:** the article number of the spray ball  
The text can be freely selected.

**Connector type:** the connector type of the spray ball  
The text can be freely selected.  
Use the connector type to distinguish between spray balls of the same type with different connectors.

Geometry

This section contains information on the geometric properties of the spray ball. This information is used for the realistic representation of the spray ball in the 3D scene.

**Height** the height of the spray ball including the connector in millimeters

**:** The value must be greater than 0 and greater than or equal to the diameter.

**Diameter:** the diameter of the spray ball in millimeters

The value must be greater than 0 and less than or equal to the height.

## Working range

This section contains the information on the working range of the spray ball. The working range describes the effective range in the nozzle direction for which the spray ball was designed. ADVISIM<sup>3D</sup> only determines a volume application or a cleaning effect if the distance between the nozzle and a point on the model surface is within the working area. No hits are displayed outside of this range.

**Maximum:** the maximum value of the working range of the spray ball in millimeters

The value must be greater than 0 and greater than the minimum of the working range.

**Minimum:** the minimum value of the working range of the spray ball in millimeters

The value must be greater than 0 and less than the maximum of the working range.

## Pressure

This section contains information on the [working pressures](#) at which the spray ball can be operated.

**Maximum:** the value for the maximum permissible operating pressure of the spray ball in bar

**um:** The value must be greater than 0 and greater than the standard pressure and the minimum pressure.

**Standard:** the value for the working pressure of the spray ball in bar

**rd:** The value must be greater than 0, less than the maximum pressure and greater than or equal to the minimum pressure.

The value specifies the typical pressure at which the nozzle is operated.

**Minimum:** the value for the minimum permissible operating pressure of the spray ball in bar

**m:** The value must be greater than 0, less than the maximum pressure and less than or equal to the standard pressure.

## Volume flow

This section contains the volume flow that the spray ball delivers at a certain pressure. These two values are required so that ADVISIM<sup>3D</sup> can calculate the corresponding volume flow for different working pressures.

**at maximum pressure:** the value for the volume flow in liters per minute that the nozzle delivers at maximum pressure

The value must be greater than 0 and greater than or equal to the volume flow at standard pressure.

**at standard pressure:** the value for the volume flow in liters per minute that the nozzle delivers at standard pressure

The value must be greater than 0 and less than or equal to the volume flow at maximum pressure.

4.11.11.2.3 Spray ball parameters

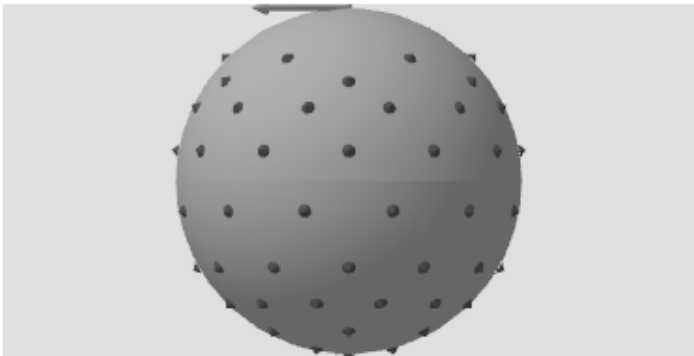
In this screen, you can enter the typical parameters for spray balls. The screen is divided into the sections described below.

Spray area

Spray direction: ☒ all around  
☐ upwards  
☐ downwards

Spray angle:  °

Spray ball preview



Hole rings

+	Latitude	Offset	Number of holes
-	45°	15°	6
-	55°	-15°	6
-	65°	0°	12

## Spray area

This section contains information on the spray area of the spray ball. The spray area indicates the general effective area of the spray ball. This information is used to display the hit area in the 3D scene.

- Spray direction:** specifies the direction in which the spray ball sprays  
One of the three options must be selected.  
The direction must be specified so that the connection is at the top when the spray ball is viewed from the side, i.e. "upwards" sprays in the direction of the connector, "downwards" sprays away from the connector.
- Spray angle:** specifies the angle in degrees at which the spray ball sprays in the specified spraying direction  
The value must be greater than 0 and less than or equal to 360°.  
If "all around" was selected for the spray direction, a value of 360° is automatically entered in the field and the field is deactivated.

## Spray ball preview

Here you can see a preview of the hole ring configurations that you have created in the [hole rings table](#). Controlling the view is identical to [controlling the 3D scene](#):

- Rotate camera:

Hold down the left mouse button on the spray ball preview and move the mouse.
- Move:

Hold down the middle mouse button on the spray ball preview and move the mouse.
- Scaling:

Point the mouse at the spray ball preview and turn the mouse wheel.  
This zooms the preview in or out.
- Reset view:

Double-click on the spray ball preview with the left or right mouse button.  
This resets the displacement and scaling, but not the camera rotation.

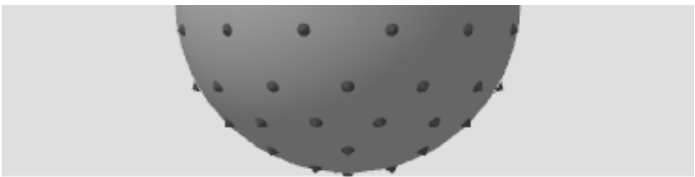
The two arrows at the top of the sphere are used for orientation and point upwards to the connector and outwards as the zero meridian.

To enlarge the spray ball preview, you can maximize the dialog window by double-clicking on its title bar and moving the window divider in the middle of the mask:

Spray angle:

360

°



Hole rings

	Latitude	Offset	Number of holes
	45°	15°	6

## Hole rings

In this table, you configure the hole rings of the spray ball. Each row in the table corresponds to a hole ring configuration.

### Adding a hole ring configuration

To add a new hole ring configuration, press the button in the table at the top left:



You can add as many hole ring configurations as you like.

### Edit hole ring configuration

To edit the values of a hole ring configuration, click on the corresponding row in the table. Text fields are then displayed that allow you to edit the individual values:

- Latitude:

specifies the latitude in degrees at which the hole ring is positioned on the spray ball  
The value must be between 0° and 180°.  
The count starts at 0° at the connector and is 90° at the equator and 180° opposite to the connector.



**Offset:** specifies the offset in degrees to the zero meridian (longitude)

The value must be between  $-180^\circ$  and  $+180^\circ$ .

You can use the offset to specify the displacement of the holes on a hole ring in relation to other hole rings. It indicates the position of the first hole of this hole ring configuration.

**Number of** specifies the number of holes on this hole ring

**holes:** The value must be greater than 0.

The positions of the holes are distributed evenly on the hole ring, starting at the position resulting from the offset to the zero meridian.

For example, a hole ring configuration with 6 holes and an offset of  $-15^\circ$  creates holes at the positions  $-15^\circ$  (corresponds to  $345^\circ$ ),  $45^\circ$ ,  $105^\circ$ ,  $165^\circ$ ,  $225^\circ$  and  $285^\circ$ , each at a distance of  $60^\circ$ .

Please note that there must be at least one valid hole ring configuration and that no hole ring configuration may be repeated.

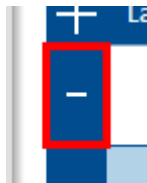
You can use the hole ring configurations to create holes at any position on the spray ball surface. For example, you can

- create individual holes that were subsequently drilled into a spray ball by creating a hole ring configuration with a hole count of 1, or
- create hole rings with uneven hole distribution by creating multiple hole ring configurations on the same latitude with different values for offset and number of holes.

The [spray ball preview](#) gives you a direct visualization of the configured hole rings. The preview is updated with every value change in the hole ring table so that you can immediately visually check the changes you have made.

## Delete hole ring configuration

To remove a hole ring configuration from the table, press the button in the header of the row with the hole ring configuration to be deleted:

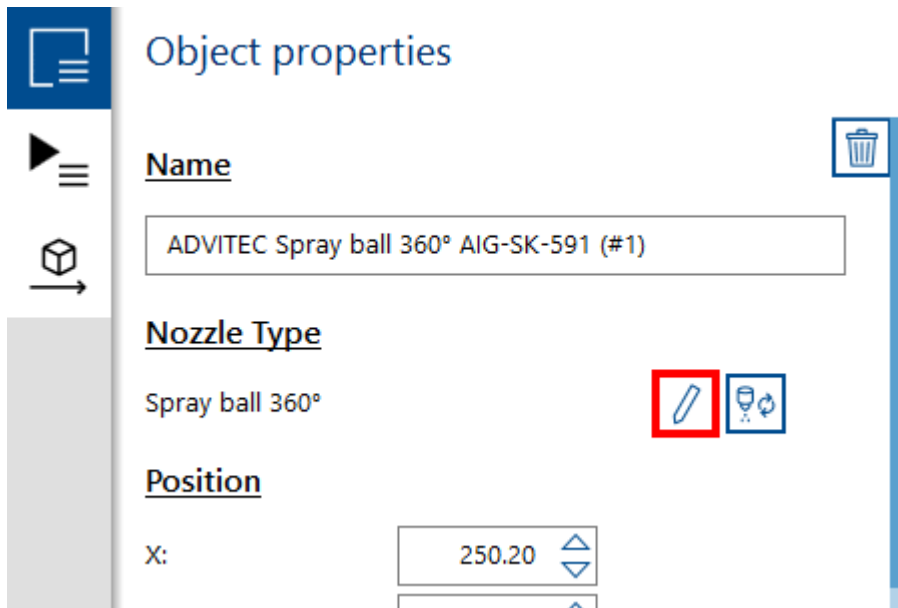


## Spray balls with hole rings pointing upwards

In the current version, ADVISIM<sup>3D</sup> does not yet support hole rings that are pointing vertically in the direction of the connector. For this type of construction, you can create a hole ring with a small value for the latitude (e.g.  $<10^\circ$ ) and use a [simulation](#) to determine whether the configuration maps the real conditions sufficiently accurate. If this is not the case, you can [edit](#) the spray ball and specify other values for the latitude in order to iteratively approach the optimum configuration.

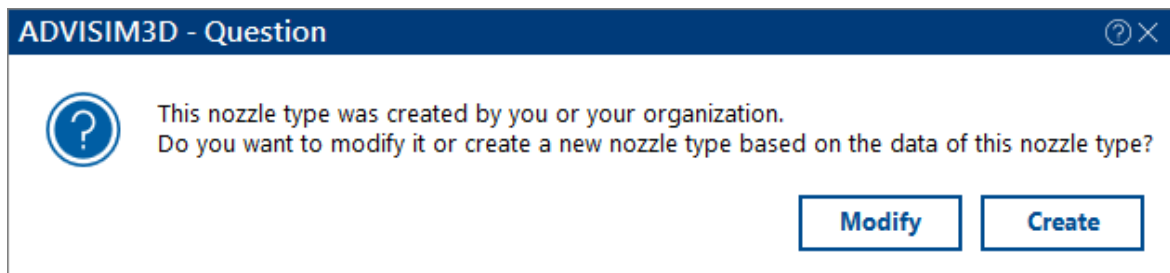
### 4.11.11.3 Edit spray balls

If you click on a spray ball in the [structure tree](#) or in the 3D scene, the button to edit nozzle types appears in the [properties area of the nozzle](#), which opens the [dialog window](#) for creating and editing spray balls.



## Editing a user-defined spray ball

If you have selected a spray ball you have created yourself and click on this button, ADVISIM<sup>3D</sup> will ask you whether you want to change the nozzle type or create a new one:



### Modify a spray ball

By clicking on the "Modify" button, ADVISIM<sup>3D</sup> displays the [dialog window](#) for creating and editing spray balls with the parameters of the selected spray ball. After accepting the data from the input masks, the spray ball is updated in the [nozzle catalog](#) and in the 3D scene. You will see the changes in the nozzle properties, the spray preview and in the simulation, depending on which parameters you have changed.

### Create a new spray ball

By clicking on the "Create" button, ADVISIM<sup>3D</sup> also displays the [dialog window](#) for creating and editing spray balls with the parameters of the selected spray ball. After accepting the data from the input masks, a new spray ball is added to the [nozzle catalog](#). The data of the spray ball selected in the 3D scene is updated.

## Editing an existing spray ball

If you have selected an original spray ball from the [nozzle catalog](#) and click on the button to edit nozzle types, no prompt appears. You create a new spray ball based on the data of the selected spray ball as described in the previous section.

#### 4.11.11.4 Delete spray balls

You can delete your user-defined spray balls. To do this, open the nozzle selection dialog and select the spray ball to be deleted. A button appears at the bottom left of the [nozzle selection dialog](#). After pressing this button, you will be asked whether you really want to delete this spray ball type. If you confirm the question, the spray ball type is removed from the nozzle catalog.

Delete custom nozzle type

*Please note:*

- If you delete a spray ball type of which nozzles are used in the current project, these nozzles will be removed.
- If you delete a spray ball type of which nozzles are used in a saved project file, these nozzles will be removed when the project file is loaded. ADVISIM<sup>3D</sup> informs you of this with a warning message.  
If simulation data was saved in the project file, it is now invalid and you must perform a simulation again.

#### 4.11.11.5 Exclusivity of custom spray balls

The spray balls you create are linked to your [ADVISIM3D license](#). Users with the same license can insert these spray balls from the nozzle catalog into their own projects or work with project files containing these spray balls without restrictions.

If you want to provide a spray ball to a user with a different license, please [contact](#) ADVISIM<sup>3D</sup> support.

## 4.12 Clipping plane interaction

Clipping planes offer the possibility to hide certain areas. You can use two different types of clipping planes in ADVISIM<sup>3D</sup>.



1. **Model-related clipping planes.** These are independent of the camera position (view / perspective) and always remain at their configured position in the 3D scene.
2. **View-related clipping planes.** These always follow the camera position (view / perspective) and therefore change their position within the 3D scene.

The [results of the simulation](#) remain unaffected. This means that model parts that are not visible due to a clipping plane are also subjected with volume flow and simulated.

You can [insert a clipping plane](#) as soon as a [model has been loaded](#).

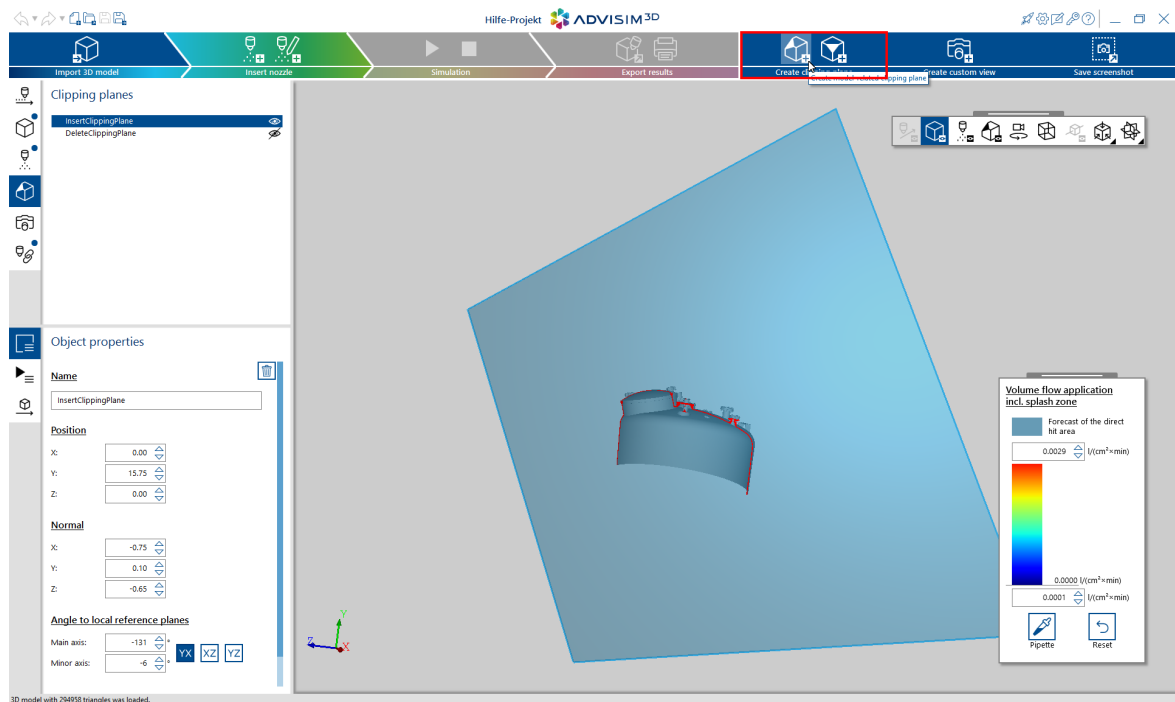
### 4.12.1 Insert clipping plane



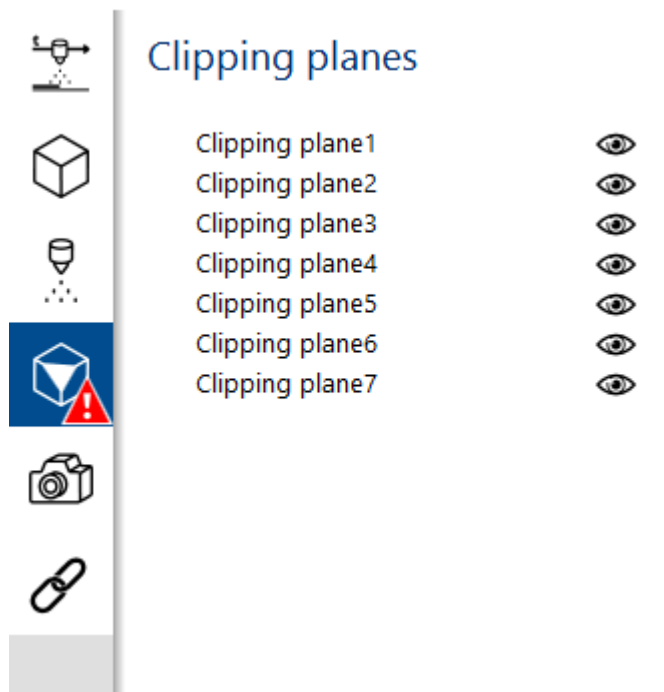
To insert a clipping plane, select the -button for a view-related clipping plane or the -button for a model-related clipping plane. You will find both buttons in the toolbar.


## 1. Model-related clipping plane

Now the clipping plane will be inserted, starting from the centre of the model and parallel to the view plane.



A maximum of 6 clipping planes can be active at the same time. If you have more than 6 active clipping planes in a project, a validation error appears, shown in the picture below, and no clipping plane is displayed in the 3D scene.



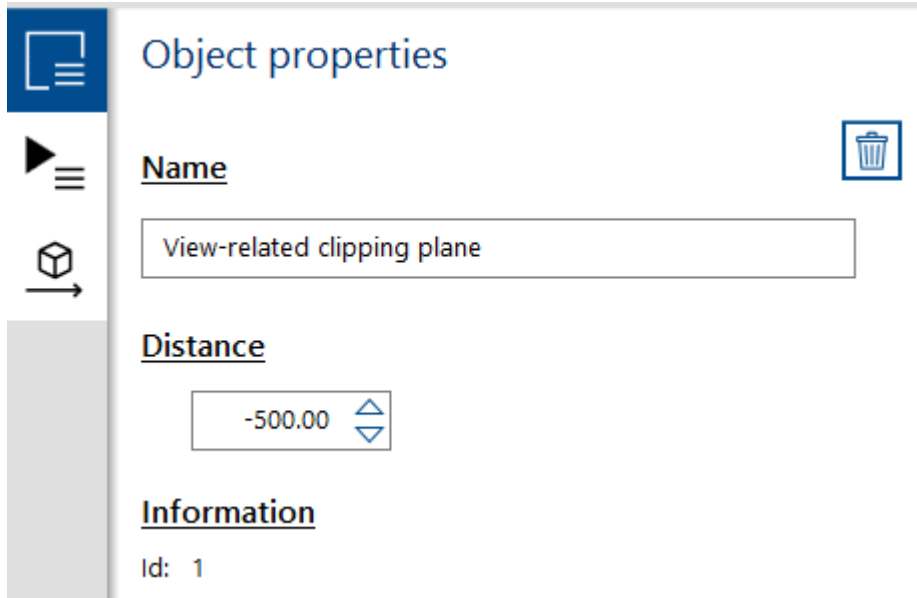
You can solve the validation error by switching off or hiding clipping planes " until you have only six active ones in the project.

Afterwards all clipping planes will be displayed in the 3D scene.

## 2. View-related clipping plane

A clipping plane, which is always parallel to the current view/camera plane is inserted, based on the centre of the model

You can use the object properties of this clipping plane to configure the distance of the plane to the centre of the model.



Object properties of a view-related clipping plane


You can insert a maximum of one view-related clipping plane. If a view-related clipping plane already exists in the project, the button is deactivated accordingly.

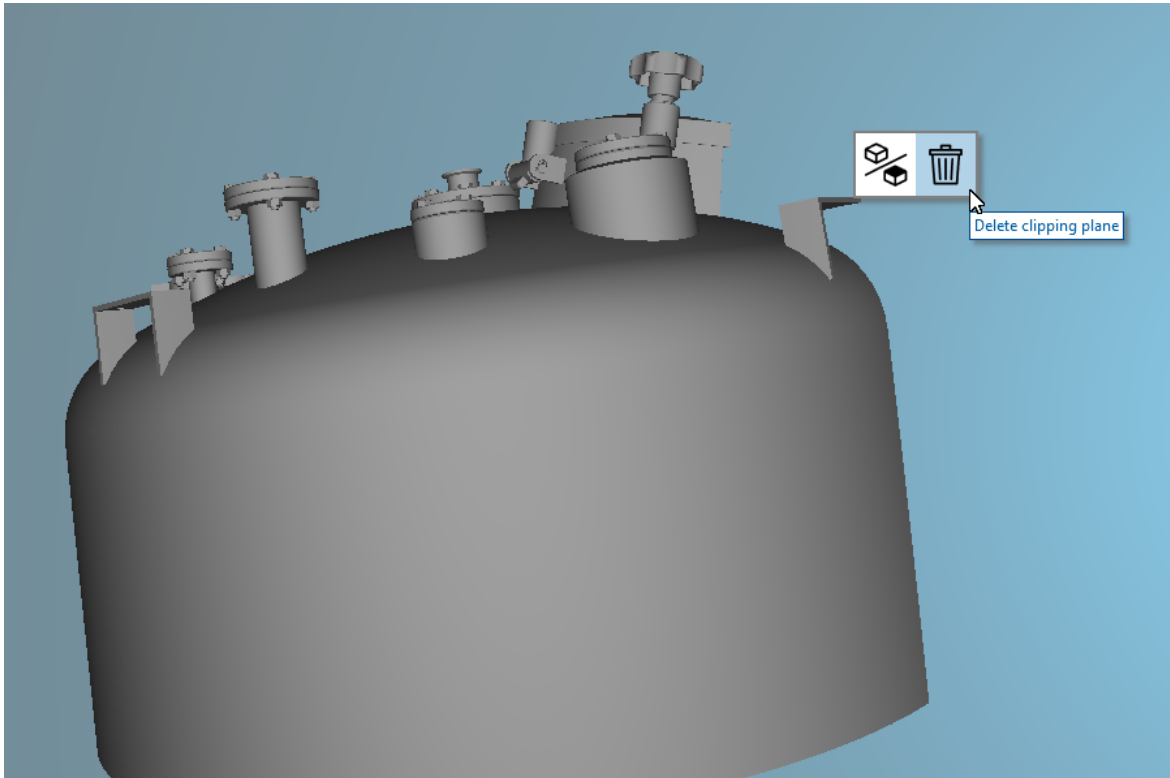
An active view-related clipping plane is also one of the maximum 6 active clipping planes that can be displayed and can therefore trigger validation errors. (See 1. Model-related section plane)

### 4.12.2 Delete clipping plane

To delete any type of a clipping plane, select the clipping plane and press the delete key.

Alternatively, you can open the [context menu](#) by right-clicking on the already selected clipping plane

and delete it by clicking the  -button.



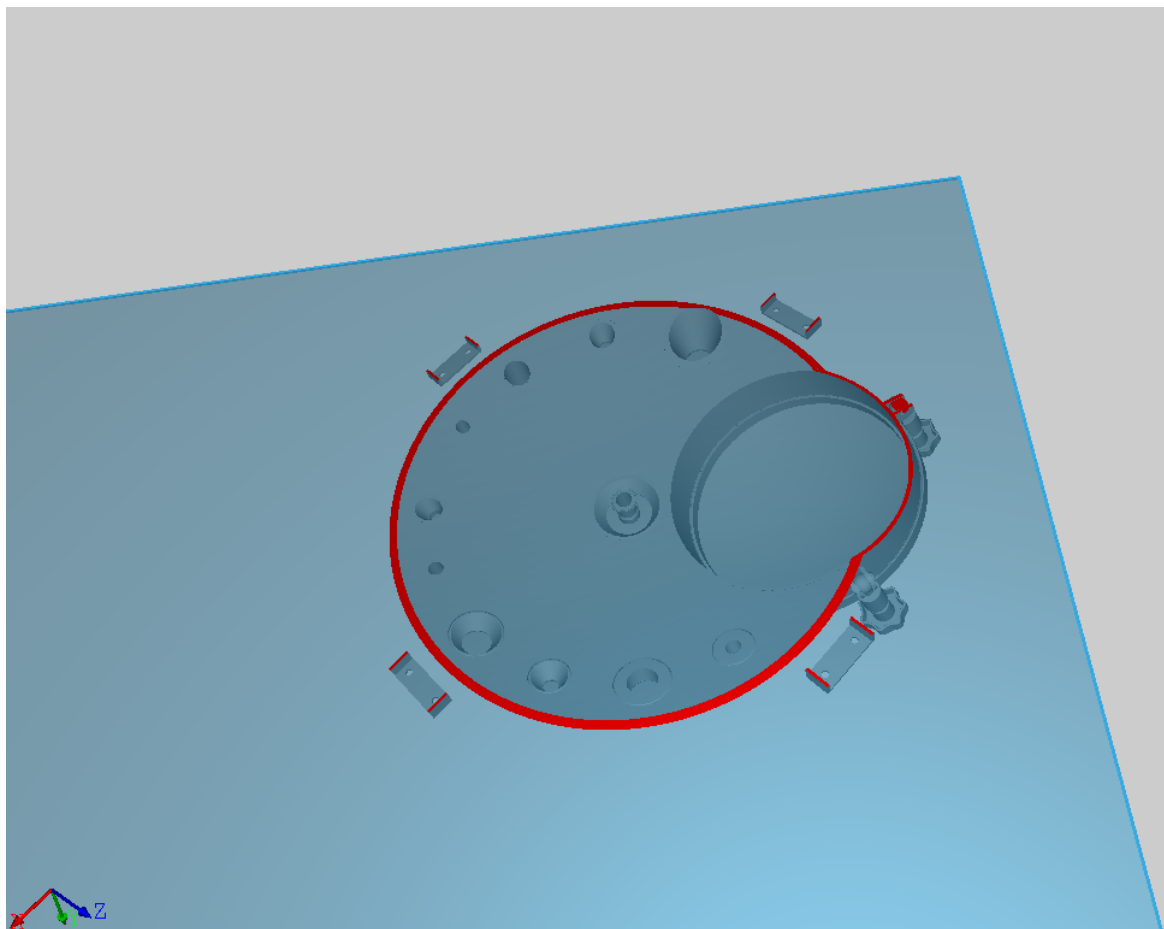
The object properties tab also contains a button for deleting the clipping plane.

#### 4.12.3 Move clipping plane along its orientation

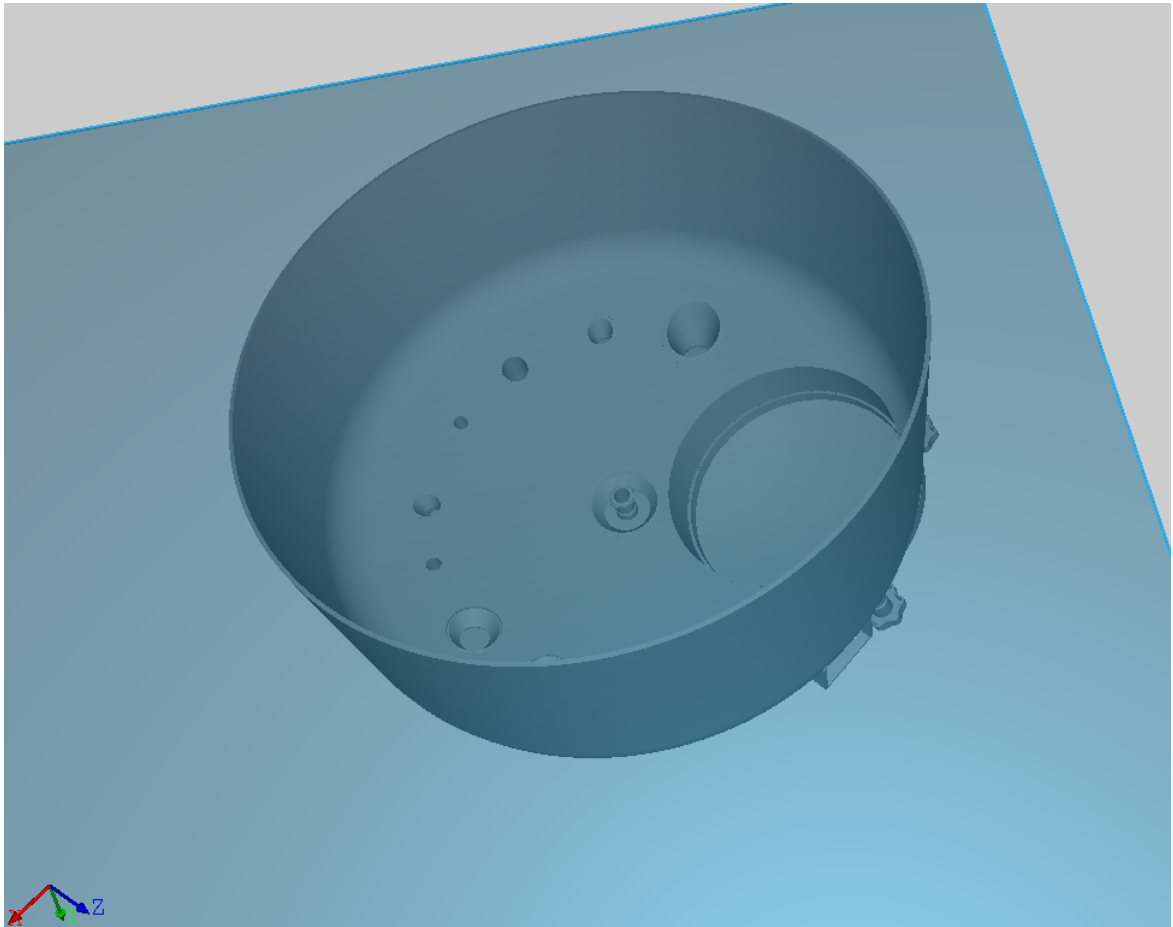
To move a clipping plane, it must first be [selected](#).

It can then be moved along its orientation by holding down the **CTRL** key and simultaneously scrolling the mouse wheel. Both types of clipping planes can be moved in this way.

Starting position:





After moving with **CTRL** + mouse wheel down



#### 4.12.4 Show and hide clipping planes

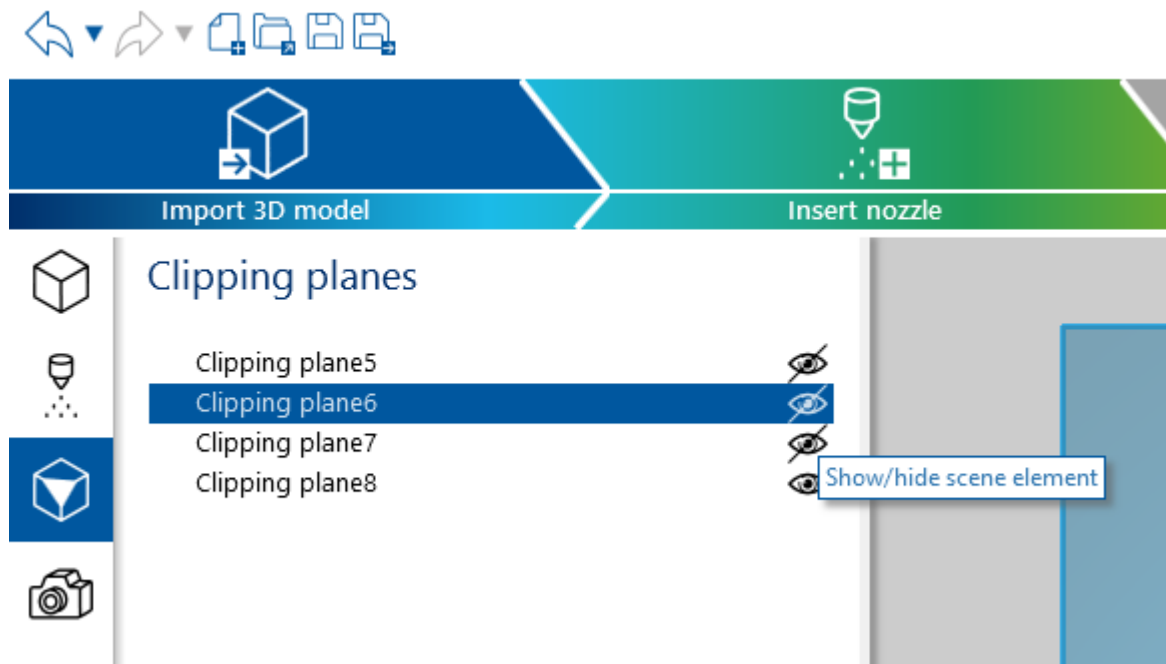
Any type of a clipping planes can be shown and hidden.

To show or hide a clipping plane, select a [clipping plane](#) and click the button  or  of the desired clipping plane in the clipping plane tab of the structure tree.

A selected and hidden clipping plane is highlighted in blue, but does not intersect the scene. The clipping plane and its state will be saved in the project file.

[The 3D toolbar contains the button to show/hide all clipping planes.](#)





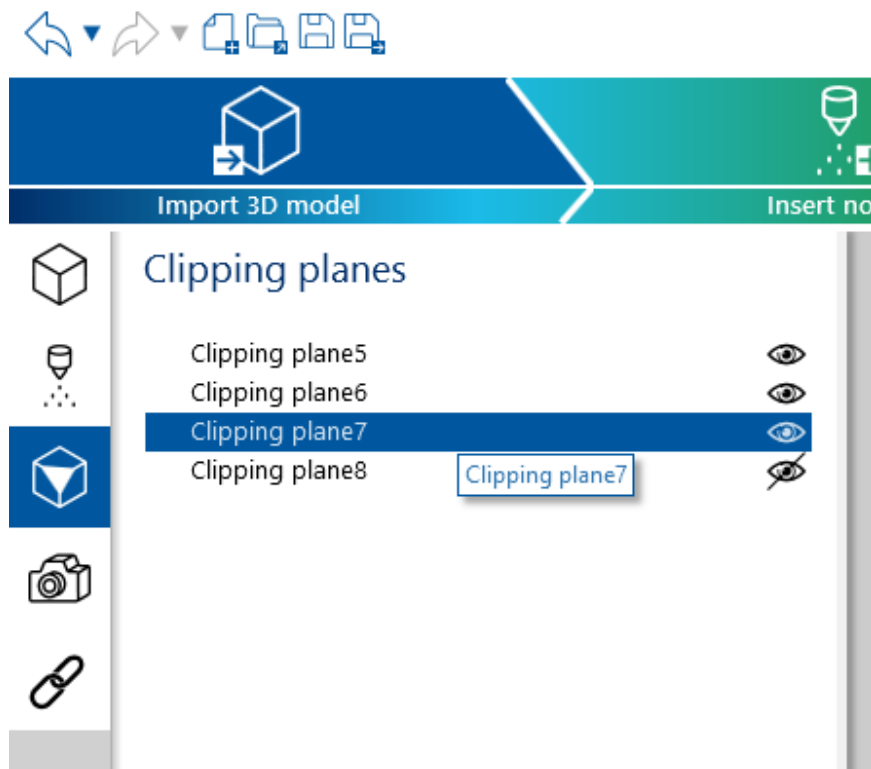
#### 4.12.5 Select clipping plane

There are two ways to select a clipping plane.

You can recognize a selected clipping plane by the fact that it is displayed in semi-transparent blue.

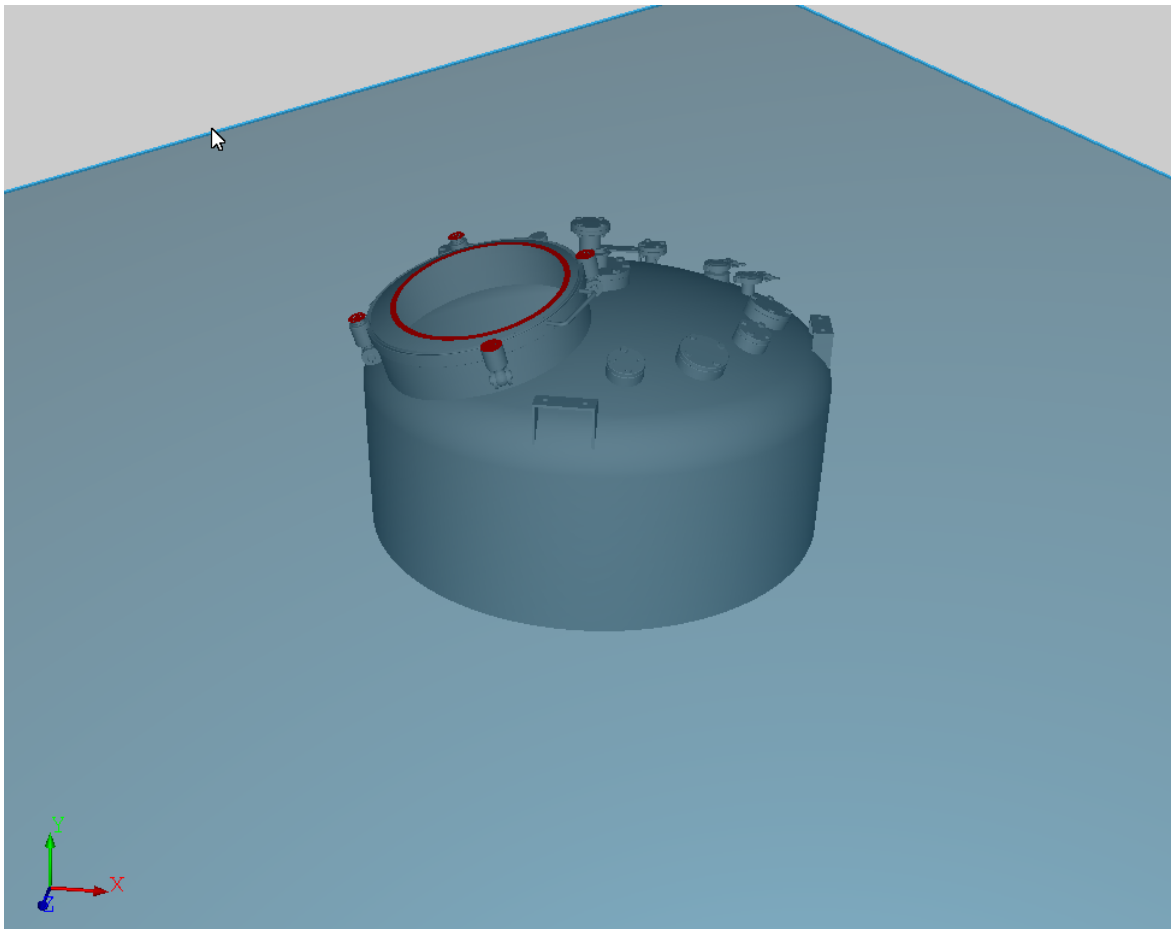
- **Selecting the clipping plane in the structure tree**

Navigate via the structure tree to the desired clipping plane and select it.




- **Selection of the clipping plane via the 3D scene**

In the 3D scene, clipping planes have a blue border that you can click on to select the clipping plane.



#### 4.12.6 Tilt clipping plane

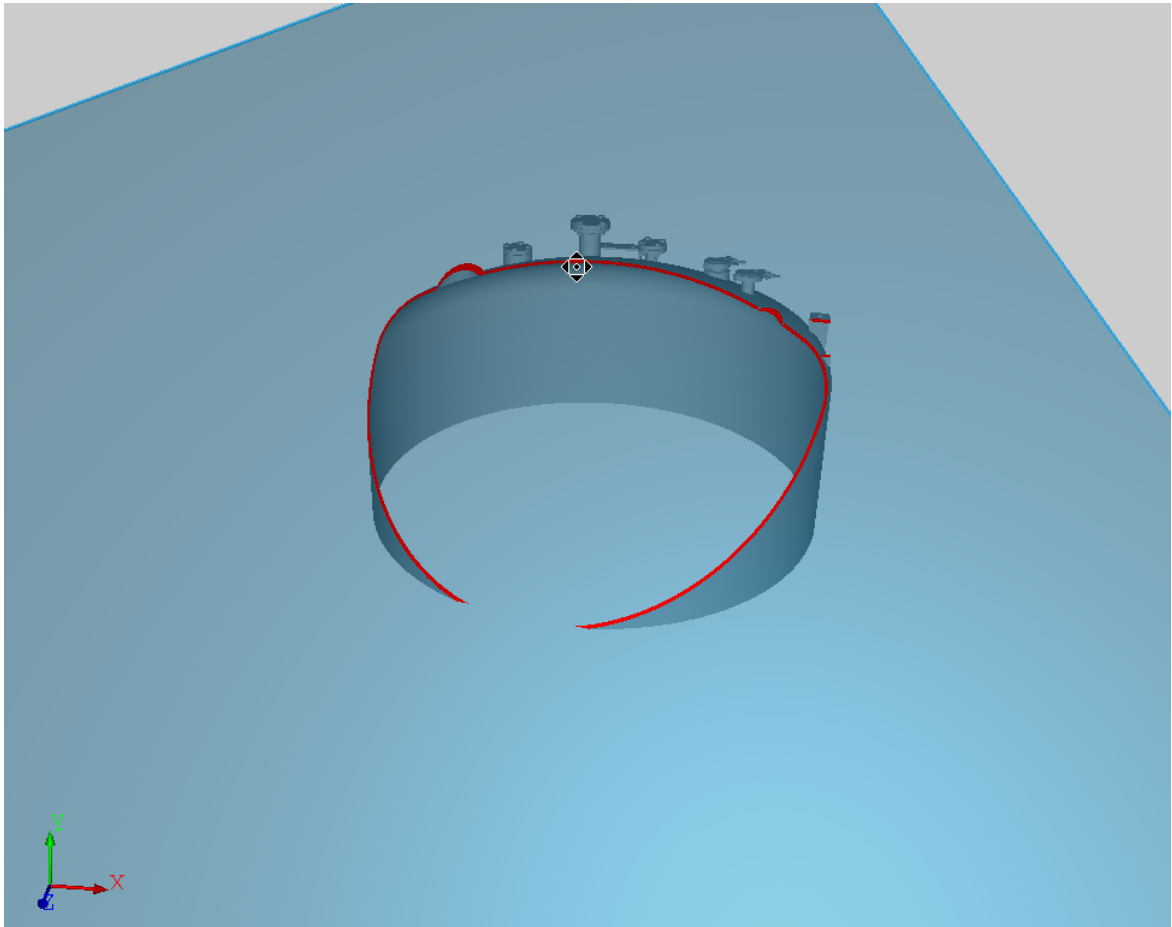
To tilt a clipping plane along the axes, it must first be [selected](#). Only model-related clipping planes could be rotated or tilted.

The mouse cursor changes to . To rotate or tilt the clipping plane afterwards, hold down the left mouse button and drag the mouse to the corresponding position. Use the **ESC** key to cancel the rotation of the clipping plane and return to the starting position and orientation.

If you drag the mouse along the displayed X-axis, the clipping plane rotates around the Y-axis.

A movement with the mouse along the Y-axis corresponds to a rotation around the X-axis.

The center of rotation is always the point at which the left mouse button was initially clicked.



## Locking individual axes

It is possible to lock individual axes so that the clipping plane rotates around only one axis.

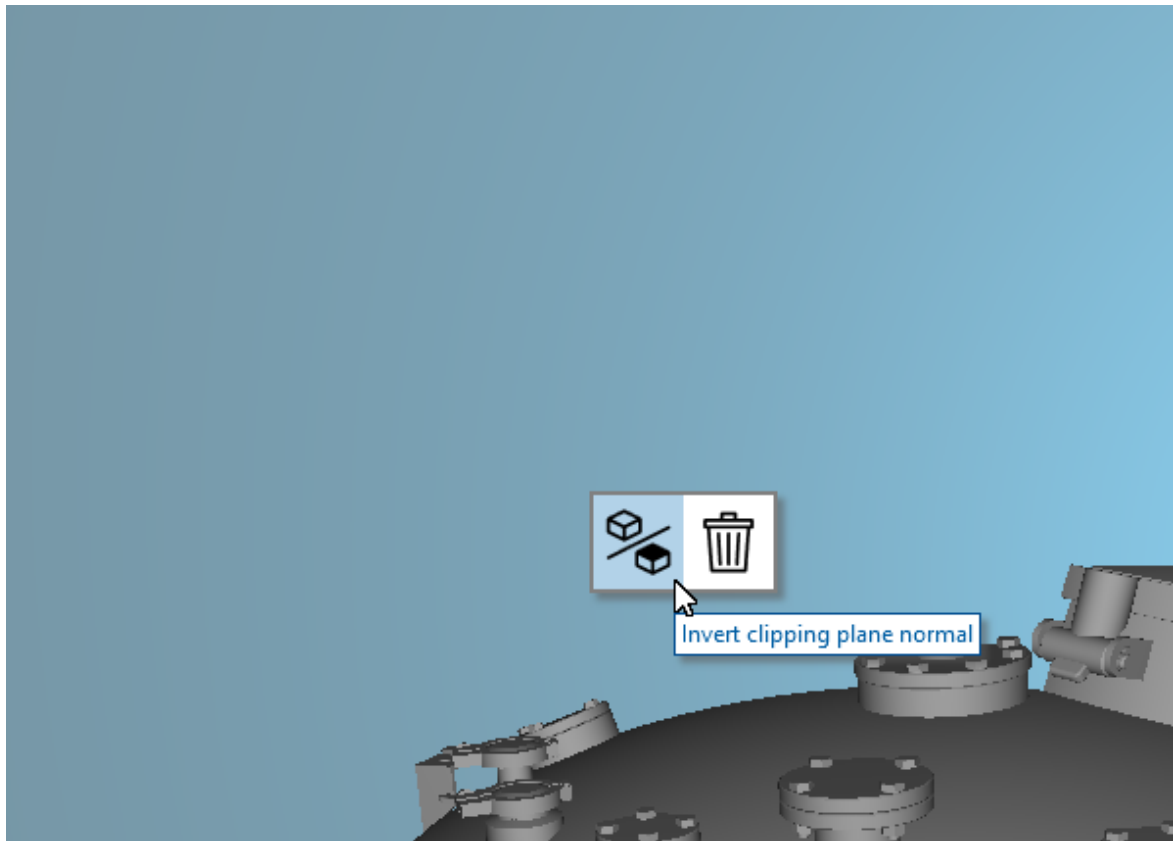
The **SHIFT** key locks the X-axis, so that the rotation is only around the displayed Y-axis.

The **CTRL** key locks the Y-axis, so that the rotation is exclusively around the displayed X-axis.

### 4.12.7 Invert clipping plane normal

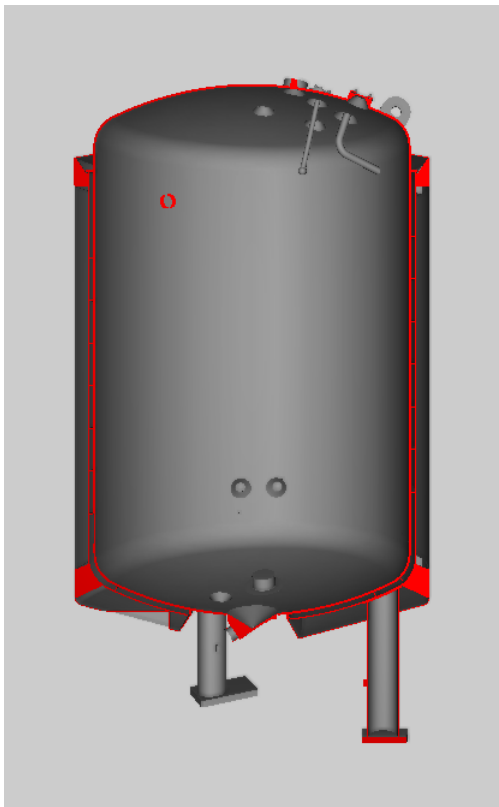
As with the [nozzles](#) it is also possible to invert the direction of the model-related clipping planes.

To do so, open the [context menu](#) and click the button *invert clipping plane normal*.



#### 4.12.8 Mark clipping planes

The cropped area of a model and clipping plane is displayed highlighted in red.



## 4.13 Constraint interaction

Constraints in ADVISIM<sup>3D</sup> offer the possibility to limit the nozzle movement in a targeted and reasonable way. Once you have inserted a nozzle, you can add one or more constraints to it.

It is possible to add several constraints to a nozzle. These are then combined with each other. All constraints applied to a nozzle count equally.

If there is a conflict with applied constraints a [corresponding error message](#) appears in the structure tree tab icon.

The conflicting constraints will also be marked in the structure tree.

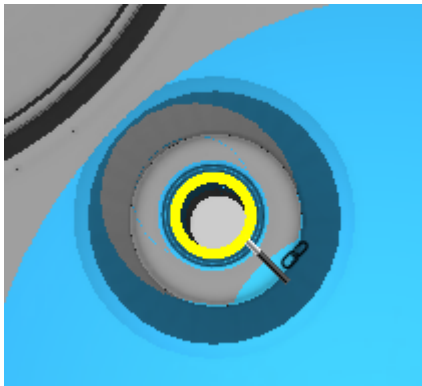
### 4.13.1 Constraint types

In ADVISIM<sup>3D</sup> it is possible to assign the following constraint types to a nozzle:

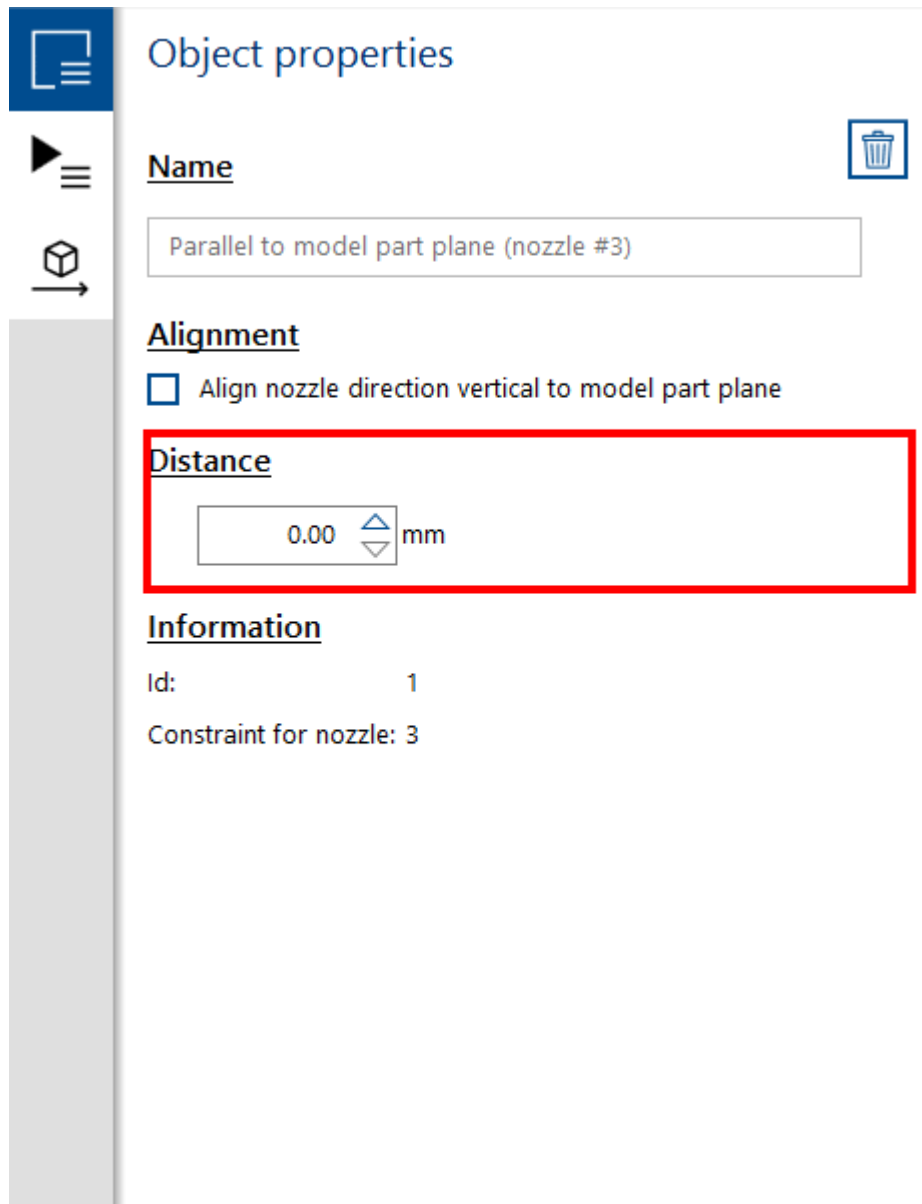
- [Parallel to a model part plane](#)
- [Along a model part axis](#)

#### 4.13.1.1 Parallel to a model part plane


If a constraint runs parallel to a model part plane, you can select any model plane. Afterwards the nozzle can only be moved on this plane.



A possible distance to the model part plane can be set via the distance value in the constraint properties.



**Object properties**



**Name** 

Parallel to model part plane (nozzle #3)

**Alignment**

☐ Align nozzle direction vertical to model part plane

**Distance**

0.00   mm

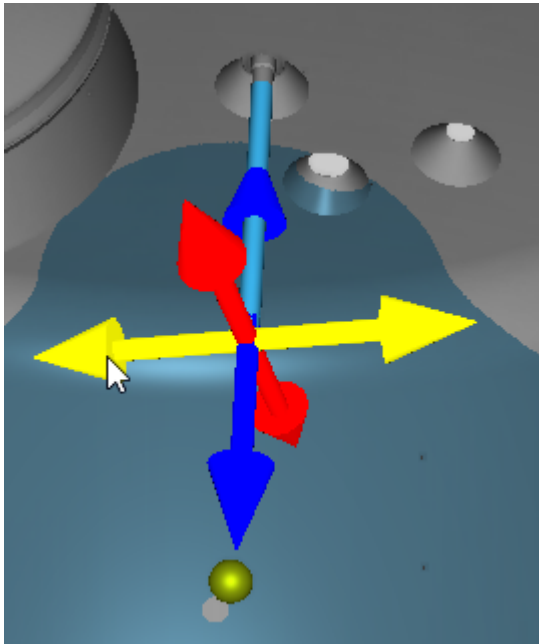
**Information**

Id: 1

Constraint for nozzle: 3

#### 4.13.1.2 Along a model part axis

When constraining along a model part axis, you can select one of the three axes for your selected model part. Afterwards the nozzle can only be moved along this axis.



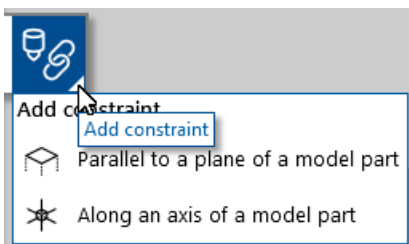
In the constraint properties you can align the nozzle orientation to the constrained axis.

#### 4.13.2 Insert constraint

To add a constraint to a nozzle, [open the context menu of the corresponding nozzle](#) and select the “Add constraints” button as shown in the picture.



In the sub-context menu that opens, you can now select which specific [constraint type](#) you would like to insert

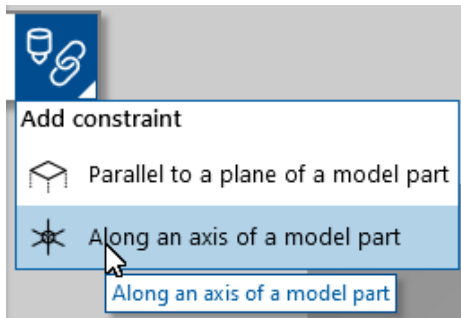


Depending on the selected type, the insertion procedure changes:

- [Add a constraint along a model part axis](#)
- [Add a constraint in parallel to a model part plane](#)

##### 4.13.2.1 Add along a model part axis

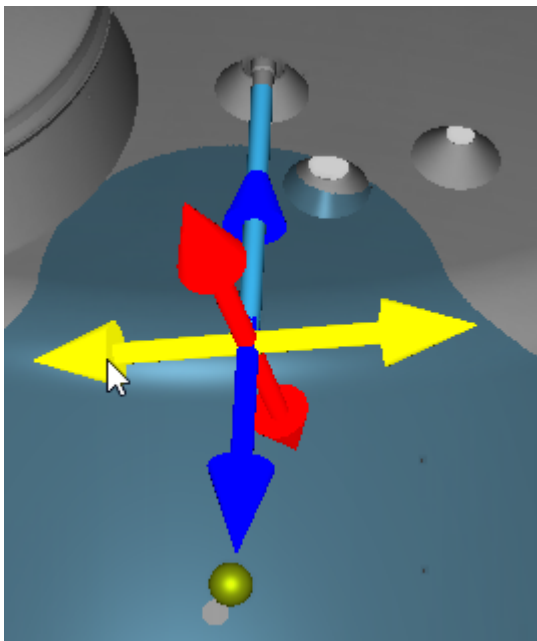
If you want to add a constraint to a nozzle along a model part axis, open the [context menu of the corresponding nozzle](#). Then go to “Along a model part axis” via “Add constraints”, as shown in the picture.



Move your mouse over the desired model part. The respective model part is highlighted in yellow. Click with your mouse on the model part to confirm.



Now the axis selection shown in the image below appears at the coordinate origin of the model part. Move your mouse over the desired axis again. Your selected axis is highlighted in yellow. Click with the mouse on the corresponding axis to confirm.



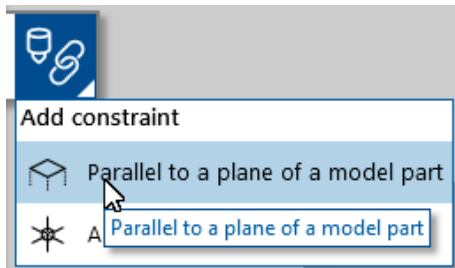
You have now successfully constrained the nozzle to the axis. The created constraint is displayed in the constraint tab in the structure tree.

By default, the nozzle direction is also constrained to the axis. You can change this due the constraint properties.

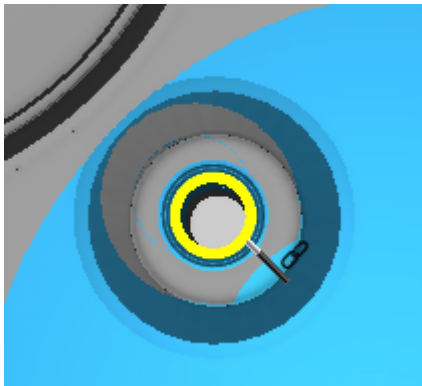


#### 4.13.2.2 Add parallel to a model part plane

If you want to add a constraint to a nozzle that is parallel to a model part plane, open the [context menu of the corresponding nozzle](#). Then go to "Add parallel to a model part plane" via "Add constraints", as shown in the picture.



Next, move your mouse over the desired area to which you want to constrain the nozzle. The respective model part area is highlighted in yellow. Click with your mouse on the desired area to confirm. The constraint can only be made for model part planes.



You have now successfully constrained the nozzle to the model part plane. In the following you can see the constraint in the structure tree.

By default, the nozzle direction is constrained to the normal of the selected model part plane. You can change this due the constraint properties.

#### 4.13.3 Error message for invalid constraint combination

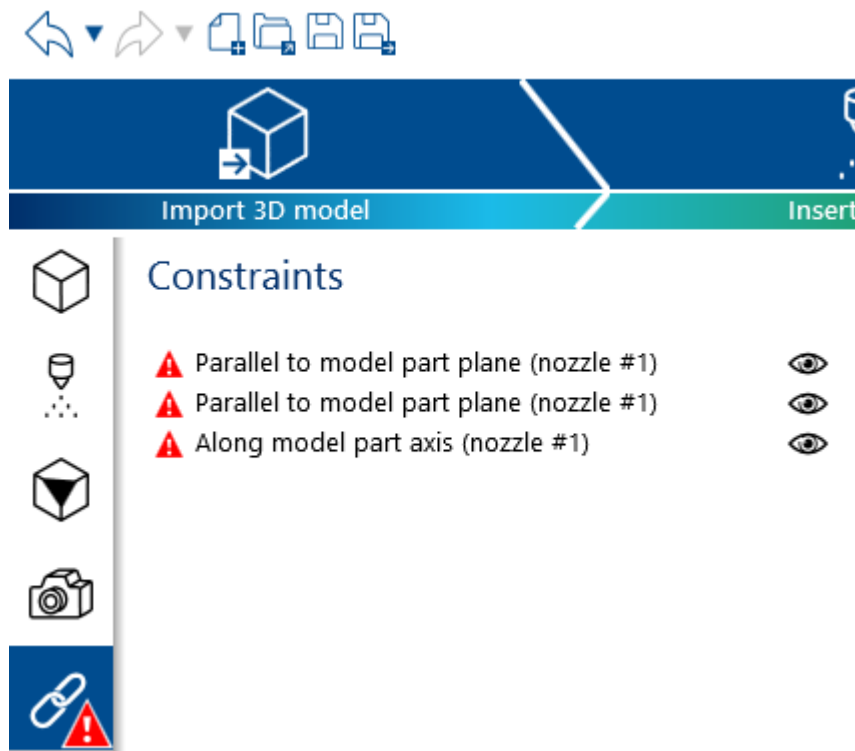
If the error message shown below appears, it means that the constraints of a nozzle are contradictory.

Constraints that are in conflict with each other will not be applied until the conflict is resolved and the warning message disappears.

Incorrect combinations of constraints can occur as follows:

- Two constraints were made along a model part axis. The axes have no intersection point.
- A constraint along a model part axis and a constraint parallel to a model part plane were made. The axis is parallel to the plane.
- Two constraints were made parallel to a model part plane. The planes are parallel to each other.

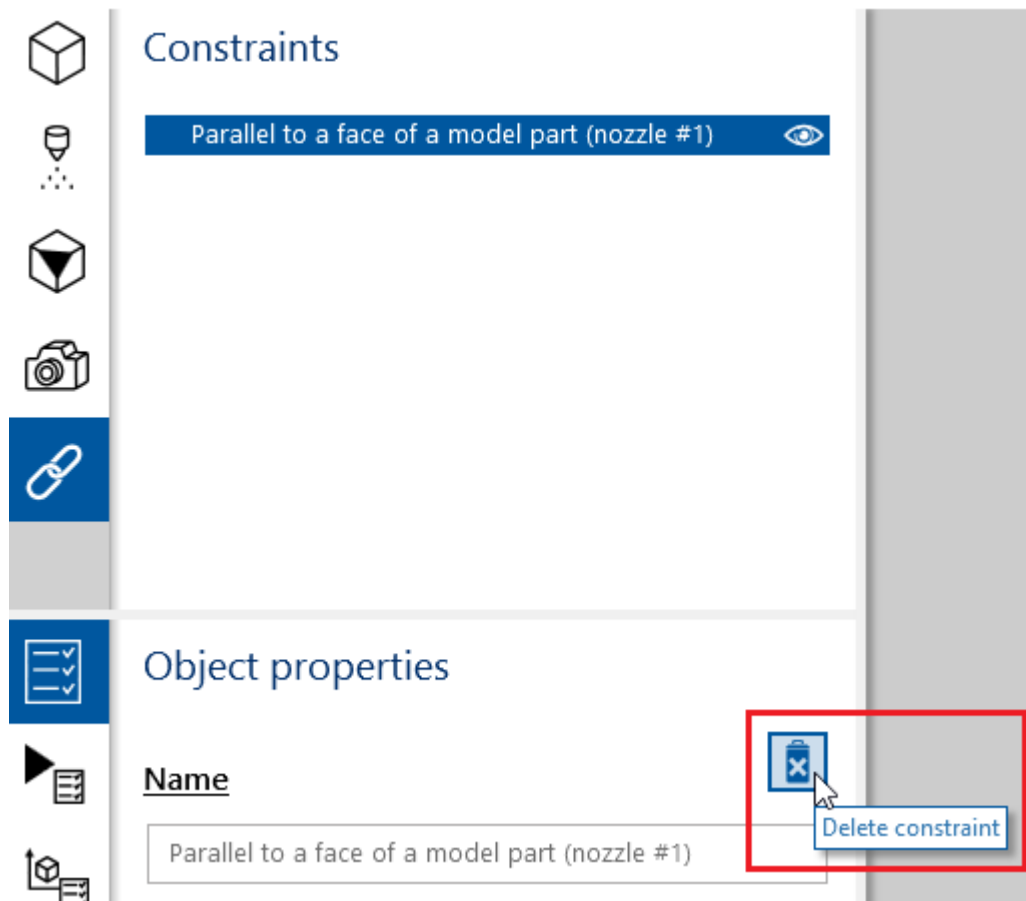
In each scenario there is no point that would fulfill both conditions.



#### 4.13.4 Delete constraint

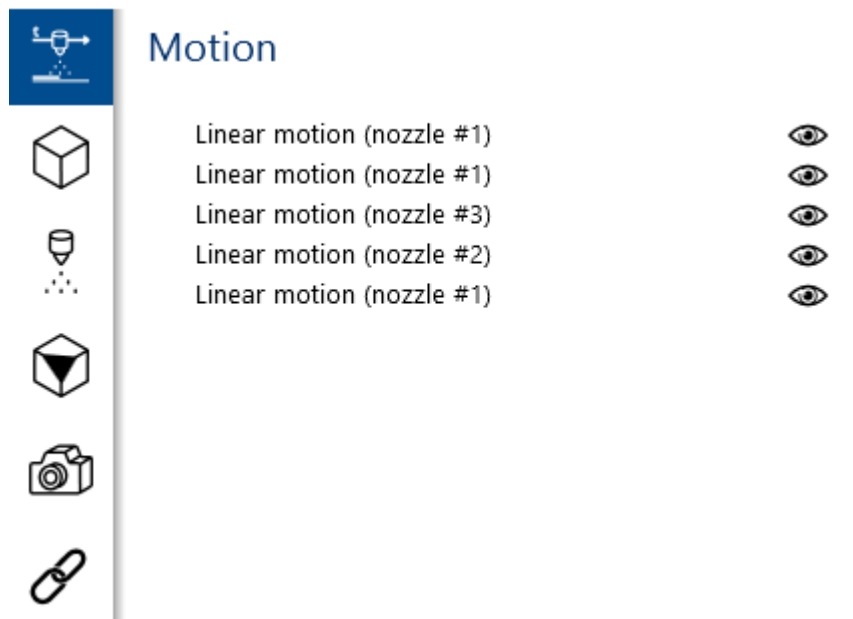
If you want to remove a constraint, please select the corresponding link in the constraint tab of the structure tree. You can now delete the constraint by pressing the delete key.

Alternatively, you can use the delete button in the properties area.



## 4.14 Motion interaction

In ADVISIM<sup>3D</sup>, you can assign and combine several motions to each nozzle in the project.



### 4.14.1 Add motion

To add a motion to a nozzle, [open the context menu of the nozzle](#) and select the "Add Motion" button as shown in the picture.



In the opened sup-context menu, please select the motion type you want to insert.

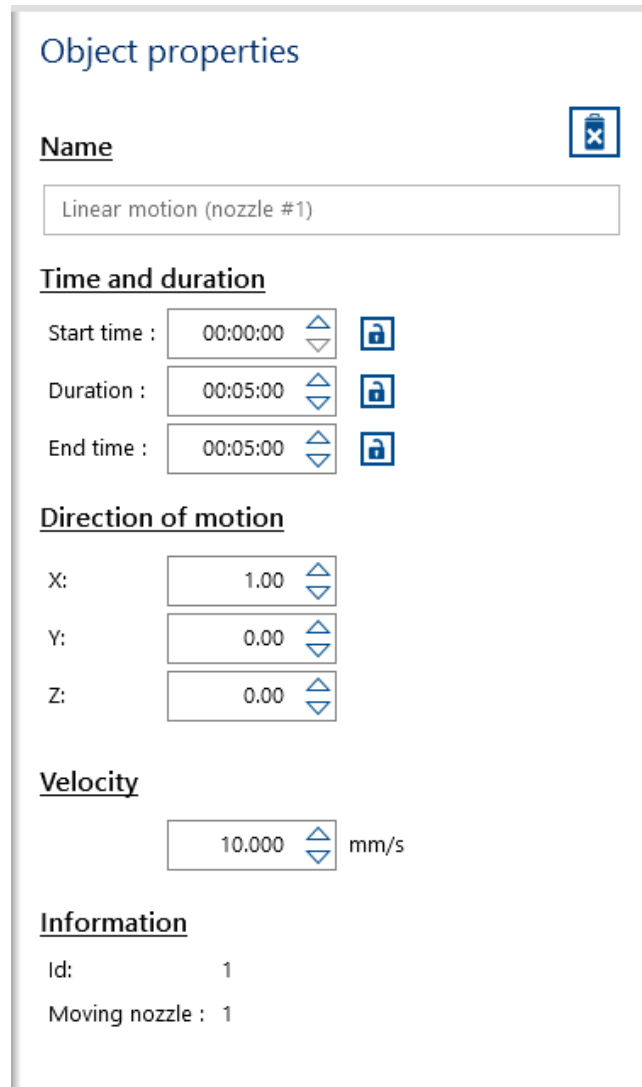
The following motion types are available:

- [Linear motion](#)
- [Rotating motion](#)

Additional motion types are currently under development.

### 4.14.2 Linear motion

You can simulate linear motions of a nozzle at a specific speed and for a specific time along a direction vector. The starting point is always the nozzle position set in the *preview*. You configure the properties and parameters of the movement using the *Object properties* tab in the Properties area of your ADVISIM<sup>3D</sup> software. To do this, you first select the motion to be set in the structure tree. In addition, you can activate and deactivate each movement here via the *eye symbol* . Deactivated motions remain unconsidered in the simulation. Changes to the properties of the linear motion can be undone with the *back arrow* and repeated with the *forward arrow* . You will find a detailed list of all properties of the linear motion in the following overview.



**Name:**

The name of the linear motion.

**Start time:**

The start time defines when the motion begins. Changing the value automatically results in adjustment of the duration or the end time. By default, first the end time is changed to its maximum (23:59:59) and then the duration. Click the lock icon to protect the entered value from changes.

**Duration:**

The duration defines how long the motion is performed. Changing the value automatically results in adjustment of the start or the end time. By default, first the end time is changed to its maximum (23:59:59) and then the start time is changed. Then click the lock icon to protect the entered value from changes.

**End time:**

The end time defines when the motion stops. Changing the value automatically results in adjustment of the duration or the start time. By default, the duration is changed first and then the start time. Click the lock icon to protect the entered value from changes.

**Direction of motion:**

The direction of motion defines the direction vector of the linear motion

**Note:** The null-vector is not supported.

**Velocity:**



Enter the velocity of the motion in mm/s.

**Note:** Negative values are not supported. To reverse the direction, please invert the direction vector.

**Id:** Sequence number of the motion. Cannot be changed.

**Moving nozzle:** Sequence number of the nozzle to which the motion refers. Cannot be changed.












### 4.14.3 Rotating motion

You can have rotating motions of a nozzle rotate around an axis at a certain speed and for a certain time during the simulation. The rotation axis is determined by the center point of the rotation and the axis vector. The starting point is always the set nozzle position in the *preview*. Configure the properties and parameters of the motion using the *Object Properties* tab in the Properties area of ADVISIM<sup>3D</sup>. To do this, first select the motion to be set in the structure tree. Use the *eye icon*  to activate and deactivate each motion. Deactivated motions remain unconsidered in the simulation. Changes to the properties of the rotating motion can be undone with the *back arrow* and repeated with the *forward arrow* icons .

You will find a detailed list of all properties of the rotating motion in the following overview.

**Name**

Rotating motion (nozzle #1)

**Time and duration**Start time : 00:00:00   Duration : 00:01:00   End time : 00:01:00   **Center of rotation**X: 0.00  Y: 0.00  Z: 0.00  **Rotation axis**X: 0.00  Y: 0.00  Z: 1.00  **Rotation velocity**1.00   rpm**Information**

Id: 1

Moving nozzle : 1

**Object properties of a rotating motion****Name:**

The name of the linear motion.

**Start time:**

The start time defines when the motion begins. Changing the value automatically results in adjustment of the duration or the end time. By default, first the end time is changed to its maximum (23:59:59) and then the duration. Click the lock icon to protect the entered value from changes.

**Duration:**

The duration defines how long the motion is performed. Changing the value automatically results in adjustment of the start or the end time. By default, first the end time is changed to its maximum (23:59:59) and then the start time is changed. Then click the lock icon to protect the entered value from changes.

**End time:**

The end time defines when the motion stops. Changing the value automatically results in adjustment of the duration or the start time. By default, the duration is changed first and then the start time. Click the lock icon to protect the entered value from changes.

**Center of rotation:**

Here you can define the center of the rotation

**Rotation axis :**

The axis of rotation defines the rotation vector of the rotating motion

**Note:** The null-vector is not supported.

**Rotation velocity:**

Enter the velocity of the rotation in rounds per minute.

**Note:** Negative values are not supported. To reverse the direction, please invert the axis of rotation.

**Id:** Sequence number of the motion. Cannot be changed.

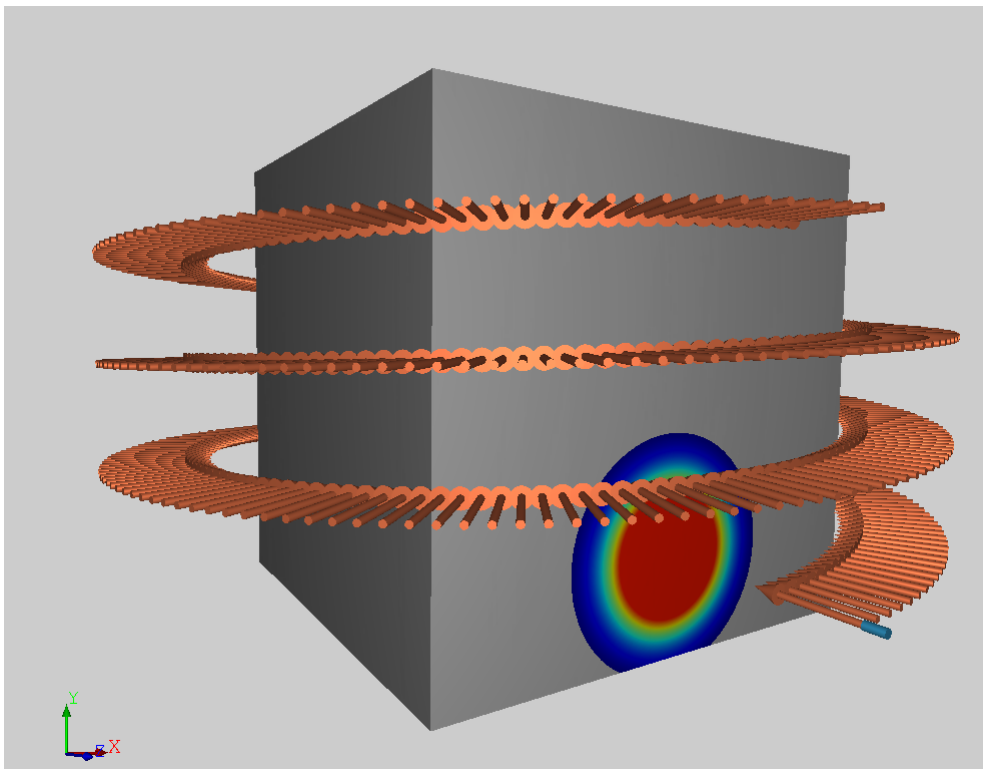
**Moving nozzle:** Sequence number of the nozzle to which the motion refers. Cannot be changed.

#### 4.14.4 Show motion path

Always keep an overview, even with several simultaneous movements, by displaying the resulting movement path of a nozzle. Click on the button in the toolbar to show or hide the movement path:



The motion path of the nozzle over time is displayed with the help of arrows to show the position and spraying direction:



The motion path can be displayed in the preview as well in the simulation view. If you change some motion parameter in the preview the motion path will be adapted immediately to the new configuration.

You can also use the time line to jump to different time steps within the configured motion.



ADVISIM<sup>3D</sup> will always display the nozzle position of the current selected time step.

Changes to the nozzle position on a selected time step will be calculated back to the start position of the nozzle at time step 0 as well to all other time steps.

#### 4.14.5 Simulate motion


With each motion inserted, the following timeline appears, enabling you to simulate motions already in the spray shadow-/preview.




Using the timeline, you can view nozzle motions over the motion time and identify configuration errors even before the final simulation.



The maximum of the timeline results from the largest end time of all motions.

Take advantage of the controls  to jump forward and backward in time steps as well as to play and pause the motion completely.


By "left-clicking" with the mouse on a point within the timeline or by dragging the slider , you will get directly to the selected time step.

Changes to the nozzle position at a currently selected time step have a direct effect on the nozzle position at all other times according to the configured motion(s).

#### 4.14.6 Delete motion

If you want to delete a motion, select it in the structure tree and delete it by pressing the **DEL** key.

Alternatively, you can use the Delete-button  in the properties area.

Deleted motions can be undone and repeated by pressing the  -button.

### 4.15 Model movement

ADVISM<sup>3D</sup> offers you the possibility to move your model in a straight line.

This allows scenarios like a conveyor belt to be simulated.

To set a linear movement, first a [model must first be imported](#).

In the [properties area](#) you will find the configuration options for [linear movement](#).

### 4.15.1 Set movement

As shown in the picture you have the following setting options:

**Model movement**

☒ Active

**Velocity**

100.00 mm/s

**Direction**

X: 1.00

Y: 0.00

Z: 0.00

**Distance to origin**

30.00 mm

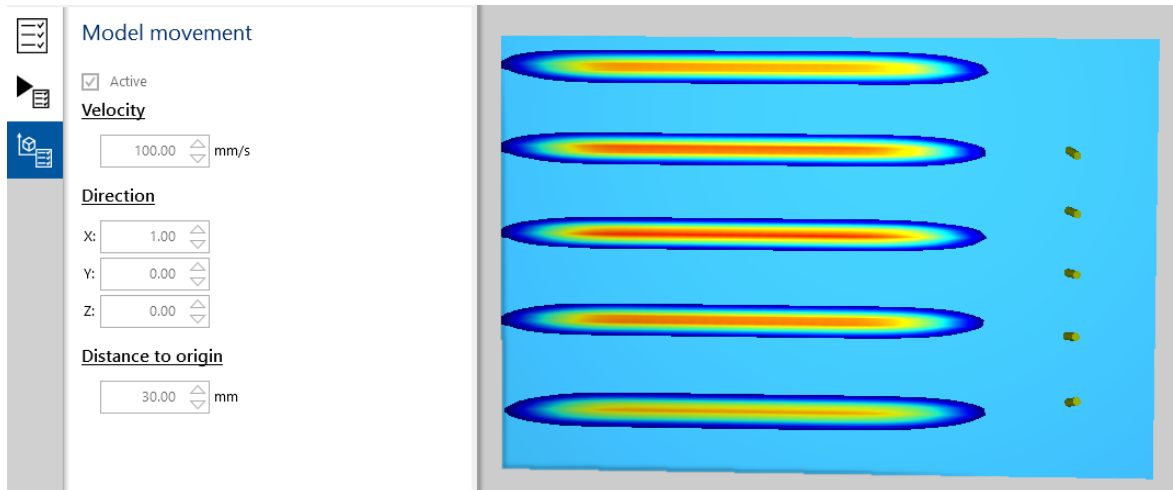
- **Active:** Activates or deactivates the movement.
- **Speed:** The speed at which the model moves along the direction is required for the [simulation component](#).
- **Orientation:** Specifies the path or direction along which the model should move from the model center. The [movement line](#) can be made visible in the 3D toolbar.
- **Distance to origin:** The distance of the model from the origin is displayed here.

### 4.15.2 Show movement line

When the movement line is switched on, you will be shown along which direction the model is moving.

The line starts in the middle of the model.





## 4.16 Custom camera views

Custom views help you to jump back to certain perspectives at any time.

The views are saved in the project and can therefore be reused at any time.

In this way, you can switch to specific problem areas and insert these views into the print report as images.

### 4.16.1 Insert custom camera views

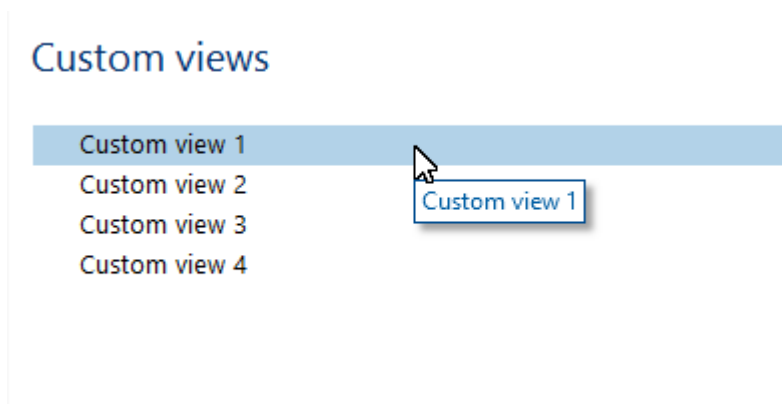
Use the Insert button in the toolbar to add the current view of the model to the project.



Both the current view and the clipping planes currently used are saved in the custom view.

## Select custom camera view

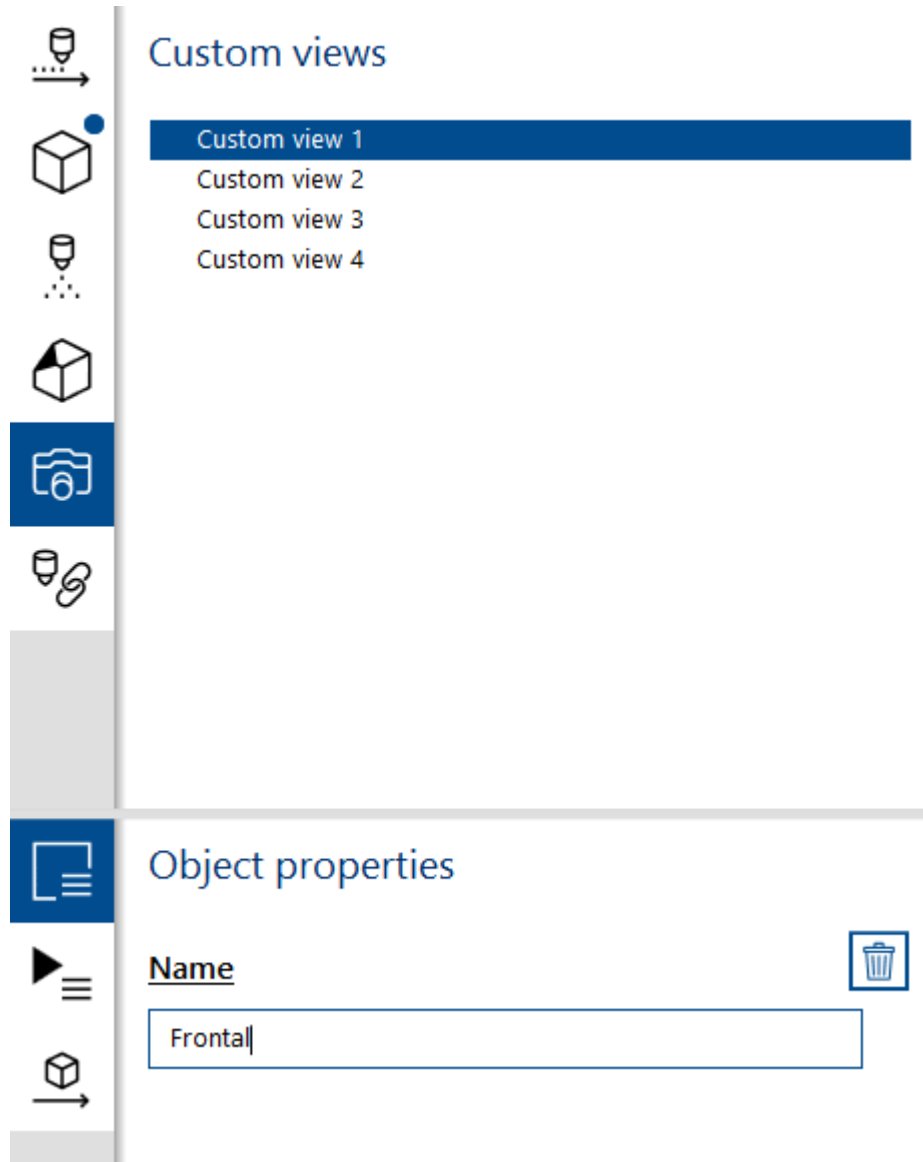
Click on the entry in the structure tree and switch back to exactly this view.



Custom views are also saved to the project file with [Save Project](#).

### 4.16.2 Change custom camera view name

Change the name of your custom view by clicking on the name in the properties area of a selected view. You confirm the name change with the **ENTER** key or by selecting another object. The new name is now displayed in the structure tree.



### 4.16.3 Delete custom camera view

You can [select](#) a view added to ADVISIM<sup>3D</sup>.

This opens up the [properties area](#) for your custom views. Click on the highlighted button to delete the view.

## Object properties

Name



Custom view 1

## 4.17 Simulation

Simulation is a core component of ADVISIM<sup>3D</sup>.

Begin [to start the simulation](#).

In the simulation, various effects are considered, which are displayed in color after the calculation of the data.

In addition to the direct impact, which is also displayed before the simulation, the following effects are also taken into account:

- Rotation of nozzles
- The cleaning effect of the draining water (if the corresponding mode is [activated](#) and [selected](#))
- The cleaning effect in the splash zone
- Detailed spray patterns are used for rotation nozzles and spray balls.

Note that [hidden model parts](#) are not considered in the simulation

For more information on how to show or hide model parts, click [here](#).

### 4.17.1 Start and continue simulation

## Start simulation

To start the simulation, you need a loaded STEP model and at least one active nozzle in the 3D

scene. Now you can click the -button in the workflow area.

This brings you to the [simulation configuration dialog](#).



Alternatively, you can do this using the [Quick Start Guide](#).

## Continue simulation

In the display with the [simulation results](#), it is possible to continue this simulation with a longer duration.



Clicking the button again opens the [simulation configuration dialog](#) again.

If you want to calculate a simulation with different parameters, [close](#) the current simulation results view and start a new simulation.

If you load a project with simulation results, it can happen that not all prerequisites for a continuation of the simulation could be provided.

The simulation can then not be continued. However, all already calculated results are available.

### 4.17.2 Stop simulation

If you have started a simulation in ADVISIM<sup>3D</sup>, you can stop it at any time.

Just click the button (see picture).



As soon as the calculations of the current step are finished ADVISIM<sup>3D</sup> immediately jumps to the [simulation results view](#). All steps calculated until then are displayed here.

**Note:** At the beginning of a simulation some mandatory steps are calculated. You cannot cancel this process.

### 4.17.3 Configure simulation

In this dialog you have the option to configure the parameters of the simulation.

Two parameters must be selected:

- **Frames per second:** Here you specify how many steps or frames per second should be simulated. The finer the interval (more frames per second/steps), the longer the calculation takes. The calculated result is more detailed. This is especially important for fast rotating cleaners.
- **Time to be simulated:** The entire time interval to be simulated. The more time you choose, the longer the calculation takes.

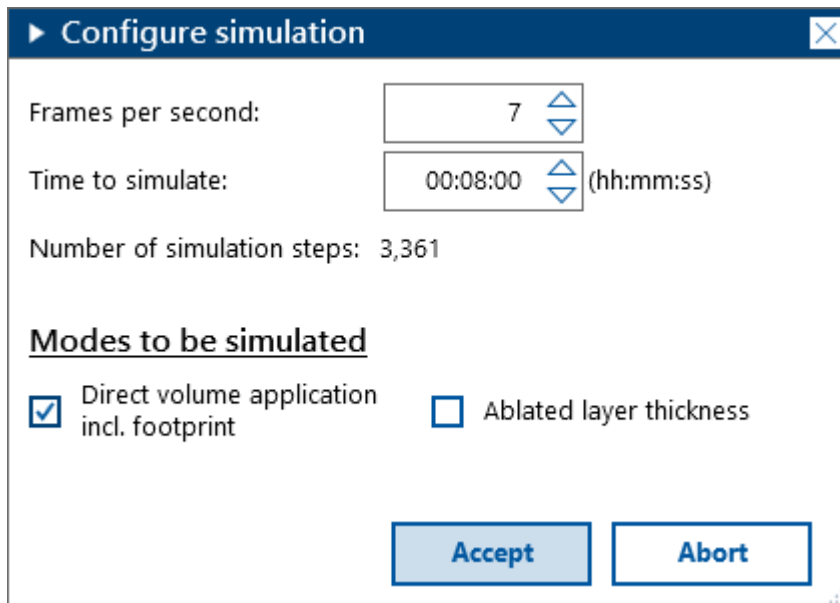
The number of simulation steps shows you how many steps the configuration you specify will contain.

Furthermore, you have the possibility to choose which [modes](#) should be simulated in ADVISIM<sup>3D</sup>.

Click "Cancel" to return to the design view.

With "Apply" you start the simulation and the individual steps are calculated.

An explanation of the individual modes can be found [here](#).



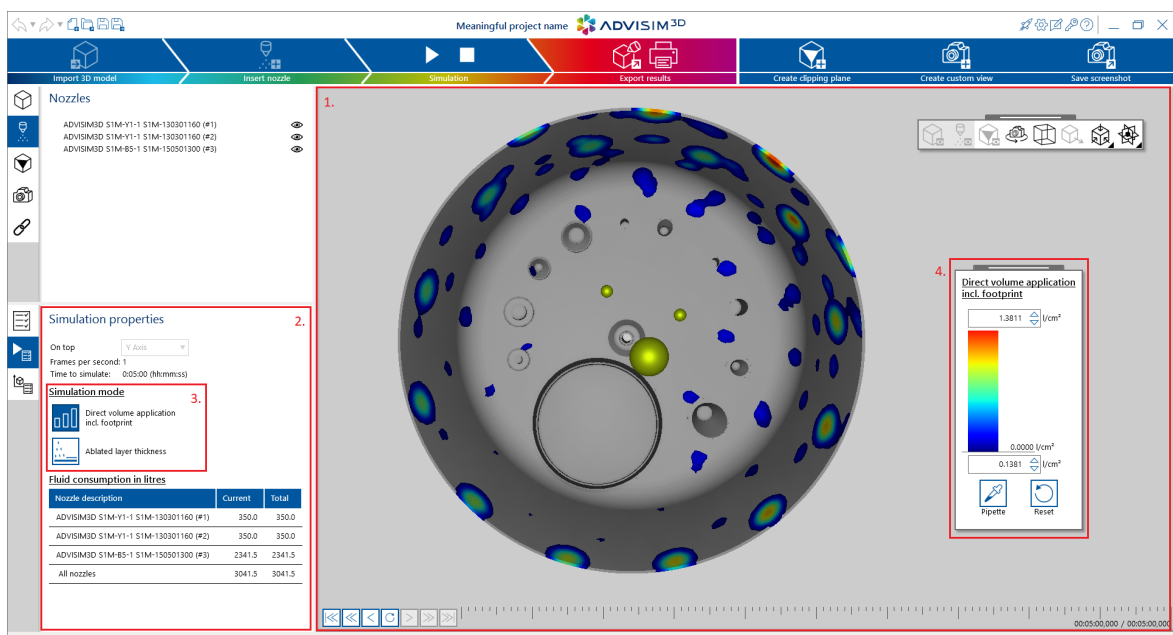
You will then be taken to the [results view](#).

When continuing a simulation only the time to be simulated can be increased. All other options cannot be changed.

#### 4.17.4 Result view

In the results view you have the following 4 components:

1. [Simulation results](#)
2. [Simulation properties area](#)
3. [Toggle the results of the simulation](#)
4. [Pipette for precise analysis of the results](#)



**Caution!** You cannot change any settings in the results view that affect the result. This includes, above all, the orientation and status of the nozzles, the import of another STEP model and the visibility of the model parts.

## Simulation results

The result of the simulation is displayed in the 3D scene. These results are displayed color-coded on the 3D model.

The [color scale](#) shows the assignment of the color value to a numerical value. You can also use the [pipette](#)

to display the exact numerical value under the mouse cursor.

A description of the values can be found [here](#).

### 4.17.5 Exit simulation

With the “Exit simulation results view” button you return to the design/construction view and can modify nozzles again.

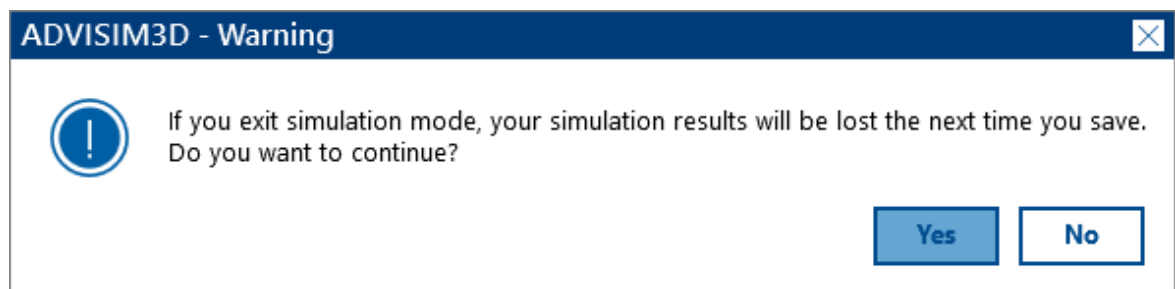


**Note:** When returning to the design mode, you will lose the simulation results.

You can first save the simulation results. A folder will then be created, named like the project file, with the suffix “.Simulation”. The simulated data from the simulation modes are stored there.

**Caution!** If you have a project with saved simulation results and return to the design mode to save them there, the simulation results will be deleted!

To save multiple simulation results, use the “[Save as](#)”-function **within** the simulation results view. Run another simulation, click “[Save as](#)” and choose a different name for the project. This creates differently named project files and their [simulation](#) folders.



### 4.17.6 Time navigation

ADVISIM<sup>3D</sup> offers you the option to view the individual steps again at a specific time and place following the calculation of the simulation results.



The control element provides the following components:

- Current time





- The total length of the simulation
- [Control elements for fast time navigation](#)
- [Control elements for running the simulation](#)

With the timeline you can move freely in time. Simply move the slider to the desired position.

#### 4.17.6.1 Playback mode




In playback mode you can watch the simulation from beginning to end. Playing back the entire simulation period always takes one minute.

The following 2 buttons are available:

-  - Plays the simulation from the selected time
-  - Pauses at current time

#### 4.17.6.2 Quick navigation

ADVISIM<sup>3D</sup> offers several buttons to control the timeline easily and accurately:

-  - Moves one interval step forward or back
-  - Goes back or forward several steps
-  - Jumps to the beginning or end of the simulation

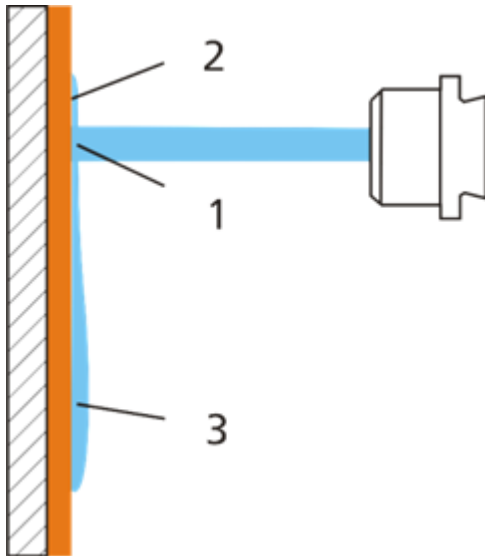
The buttons for one or more steps can be kept pressed.

#### 4.17.7 Simulation modes

In the following section you will find an overview of the modes which can be simulated in ADVISIM<sup>3D</sup>:

- [Direct volume application including footprint](#)
- [Ablated layer thickness](#)

ADVISIM<sup>3D</sup> generally distinguishes between 3 areas for the simulation:



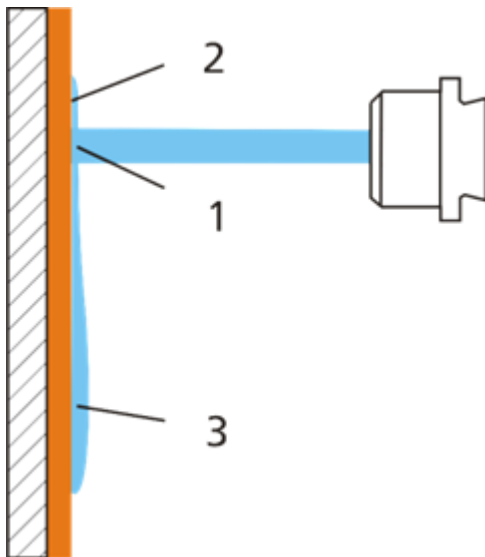
1. Direct volume distribution of the nozzle
2. Area of the nozzle footprint
3. Cleaning effect of the outflowing cleaning fluid in relation to a standard contamination

#### 4.17.7.1 Direct volume application including footprint

In this mode it is displayed how high the water column is that has accumulated on this plane during the entire cleaning process.

The unit displayed on the color scale indicates the applied water column over the total running time of the simulation.

For this mode the effects of the [volume distribution\(1\)](#) and the [footprint \(2\)](#) are considered.



#### 4.17.7.2 Ablated layer thickness

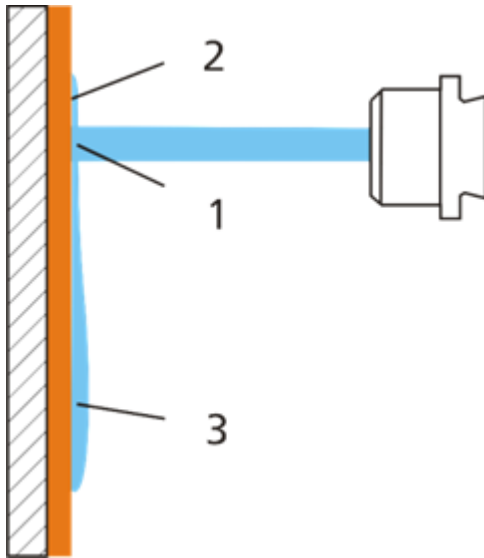
The simulated cleaning effect takes into account, among other things, the outflowing water. Altogether a statement is to be made about the extent to which a **standard contamination** can be removed from the physical conditions.

For a correct calculation it is necessary that the [top-direction](#) is set accordingly.

The unit shown in the color scale indicates the amount of contamination removed, relative to a standard contamination, in  $\mu\text{m}$ .

For this mode all effects are considered ([volume distribution \(1\)](#), [footprints \(2\)](#), [cleaning effect of the outflowing cleaning fluid \(3\)](#)).

**Caution!** Due to limitations in the experimental setup, the cleaning effect is only simulated within a radius of 1m from any direct volume flow application.



## 4.18 Print report

You have the possibility to prepare your current results as a print report. To do so, select either the print report button in the workflow area or in the [Quick Start Guide](#).



Then the configuration window shown below opens.

**Configure print report**

**Page settings**

Print format: A4 ☐ Portrait ☒ Landscape

Page margins (cm):  
 Left: 2.5 Top: 2.0 Right: 1.5 Bottom: 1.0

**Logo**

☒ Use your custom logo

Logo for cover page: C:\ADVISIM3D\Report\cover.jpg

Logo for content pages: C:\ADVISIM3D\Report\details.jpg

**General information**

Title: ADVISIMulation

Editor: Max, Mustermann

Model name: Klöpperboden\_214.STEP **File path**

☒ Description: ADVISIM3D-Simulation

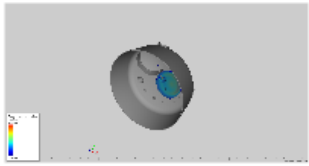
**Use information for print report**

☒ Nozzle information

☒ Hidden model parts

☒ Simulation information

**Choose custom views for report**

<input type="checkbox"/>	Name of view	Print size	Preview
<input type="checkbox"/>	InsertNozzle	Full page	
<input type="checkbox"/>	InsertClippingPlane	Full page	
<input checked="" type="checkbox"/>	Compass	Full page	
<input type="checkbox"/>	InsertNozzleFull	Full page	

**Show preview** **Print** **Cancel**

Here you have a number of settings to choose from:

- [Page settings](#)
- [General information](#)
- [Nozzle information](#)
- [Non-visible model parts](#)
- [Simulation information](#)
- [Choose 3D views for report](#)

Once you have chosen your settings, you can preview the report or print it out. Select the corresponding buttons to do so.

### 4.18.1 Page settings

In this area you can choose general print settings such as the print format or page margins or the logos you want to use.


#### Page settings


Print format: A4 Portrait ☒ Landscape

Left	Top	Right	Bottom
2.5	2.0	1.5	1.0

#### Logo

☒ Use your custom logo

Logo for cover page: C:\ADVISIM3D\Report\cover.jpg 

Logo for content pages: C:\ADVISIM3D\Report\details.jpg 

If you activate customized logos the specified logos will be added to the print report. If not or no valid path is selected the ADVISIM<sup>3D</sup> logo will be added instead. The cover page logo is scaled to a height of 2cm with a maximum width of 5cm. The logo for the detail pages is scaled to a height of 1cm.

### 4.18.2 General information

In this area you can add additional information to your print report, such as a title, the editor, the model name, or an optional description.

#### General information

Title: ADVISIMulation

Editor: Max, Mustermann

Model name: Klöpperboden\_214.STEP File path

☒ Description: ADVISIM3D-Simulation

### 4.18.3 Nozzle information

In this area you can select whether the nozzle information should be used in the print report. To do this, check or uncheck corresponding option.

### Use information for print report

- ☒ Nozzle information
- ☒ Hidden model parts
- ☒ Simulation information

#### 4.18.4 Non-visible model parts

In this area you can select whether non-visible model parts should also be used in the print report. Check or uncheck the corresponding options.

### Use information for print report

- ☒ Nozzle information
- ☒ Hidden model parts
- ☒ Simulation information

#### 4.18.5 Simulationinformation


In this area you can select whether the simulation information should be used in the print report. To do this, check or uncheck corresponding option.

### Use information for print report

- ☒ Nozzle information
- ☒ Hidden model parts
- ☒ Simulation information

If you activate the simulation information a page like the one shown below will be added to the print report.

ADVISIMulation



### Simulation information

**General parameter**

Updirection

Y-Axis

Frames per Seconds

1

Duration

00:01:00


Current Simulationtime

00:01:00,000

**Liquid consumption of the nozzle(s) in liters**

No.	Nozzle	Current	Overall
1	InsertNozzleFull	9.19	9.19
2	All nozzles	9.19	9.19

Simulation information

powered by  ADVISIM 3D

Page 5 of 6

4.18.6 Choose 3D views for report

In this area you select which of your custom views you want to include in the print report. The display ([screenshot](#)) of your custom views always refers to the current mode you are in and the current timestep.

Choose custom views for report

☐

Name of view

Print size

☐

InsertNozzle

Full page

☐

InsertClippingPlane

Full page

☒

Compass

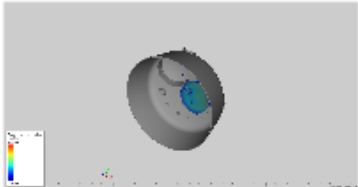
Full page

☐

InsertNozzleFull

Full page

Preview



Select the views you want to include with one click. You will see a corresponding preview in the area on the right. You can also select the later print size.

4.19 Settings

Click the button with the gear wheel in the title bar.

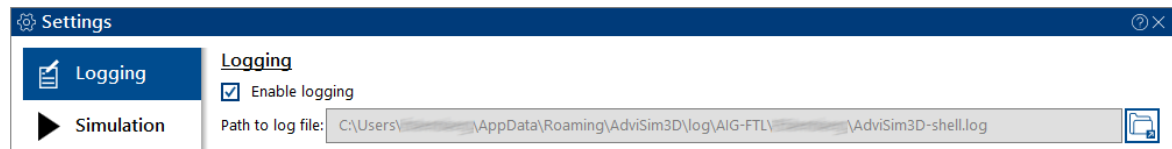


The settings dialog opens.

Within the Logging tab you can specify whether program actions should be logged.

## Logging

**Caution!** The log file can quickly become very large during intensive program use and affect program performance.



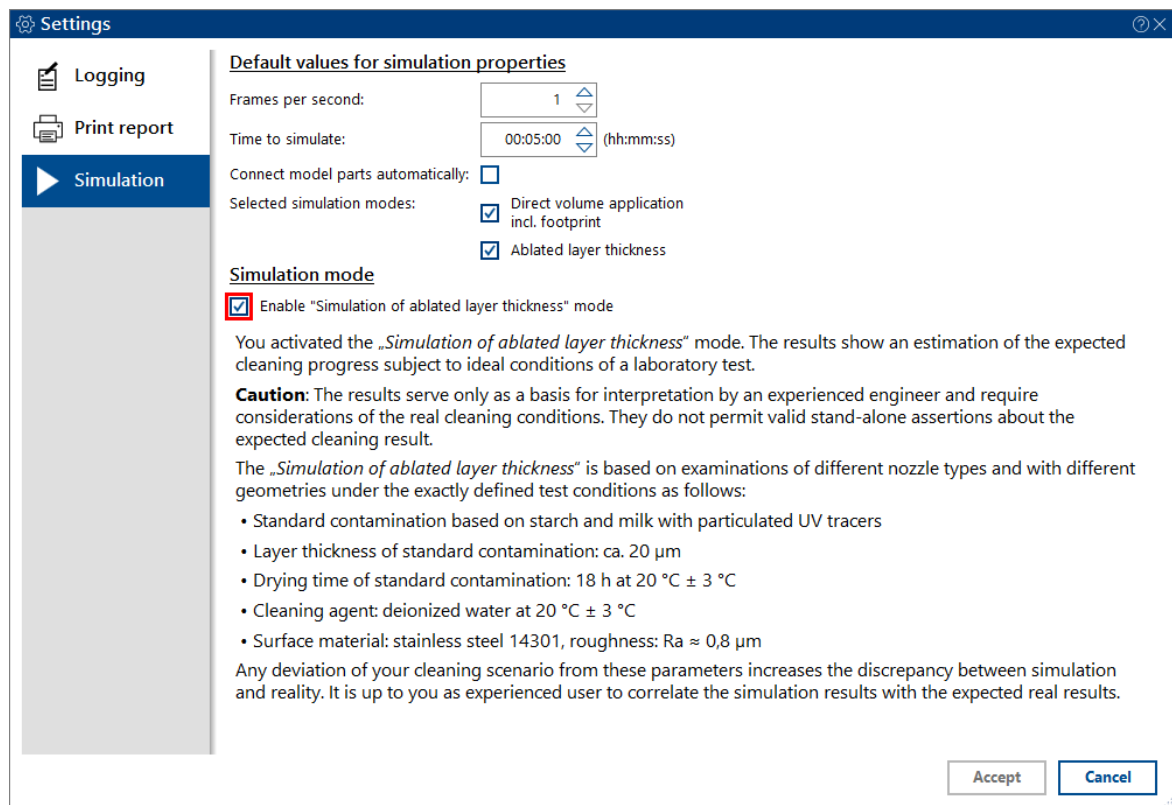
You can also click the “Open path to log file” button to display the path to the log file.

## Print report

[ERGÄNZEN]

## Simulation

Within the Simulation tab you can activate or deactivate the "determination of connected model parts" as well the "Simulation of the ablated layer thickness" mode.



Please note the following when activating the “Simulation of the ablated layer thickness” mode:

The results show an estimation for the expected cleaning process under the ideal conditions of a laboratory test.



**Caution!** The results serve exclusively as a basis for interpretation by an experienced engineer under consideration of the actual cleaning conditions and by itself do not provide a valid statement about the expected layer ablation.

The “simulation of the ablated layer thickness” is based on tests with various nozzles and on different geometries under the following exactly defined test conditions:

- Standard contamination on starch-milk basis with particulate UV tracer
- Layer thickness of standard contamination: approx. 20 µm
- Drying time of standard contamination: 18 h at 20 °C ± 3 °C
- Cleaning medium: demineralized water 20 °C ± 3 °C
- Surface: stainless steel 14301, roughness: Ra ~ 0,8 µm

Any deviation of your specific cleaning scenario from these parameters increases the discrepancy between simulation and reality. It is up to you as an experienced user to establish the correlation between the simulation results and the real, actually expected results.

#### 4.19.1 Logging

Geben Sie hier den Text ein.

#### 4.19.2 Print report

Geben Sie hier den Text ein.

#### 4.19.3 Simulation

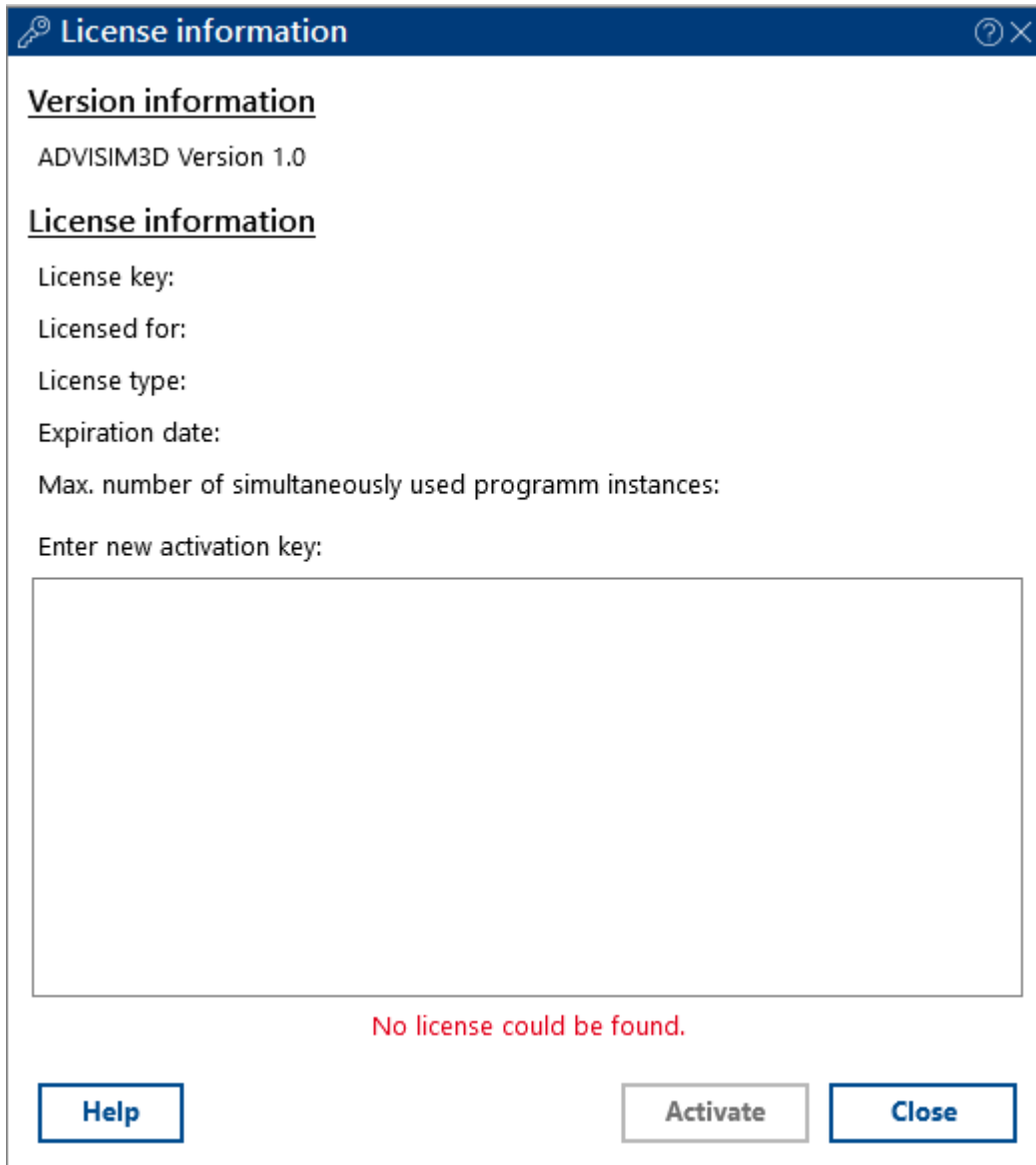
Geben Sie hier den Text ein.

### 4.20 License management

Here you will learn how to [activate](#) ADVISIM<sup>3D</sup> and display [information about your license](#).

### 4.20.1 Activate license

When you start ADVISIM<sup>3D</sup> you are asked to enter a license key.



The image shows a 'License information' dialog box with a blue title bar. It contains two sections: 'Version information' showing 'ADVISIM3D Version 1.0' and 'License information' with labels for 'License key:', 'Licensed for:', 'License type:', 'Expiration date:', and 'Max. number of simultaneously used programm instances:'. Below these is a large text input field for the 'Enter new activation key:'. At the bottom, there is a red error message 'No license could be found.' and three buttons: 'Help', 'Activate', and 'Close'.

**License information**

Version information

ADVISIM3D Version 1.0

License information

License key:

Licensed for:

License type:

Expiration date:

Max. number of simultaneously used programm instances:

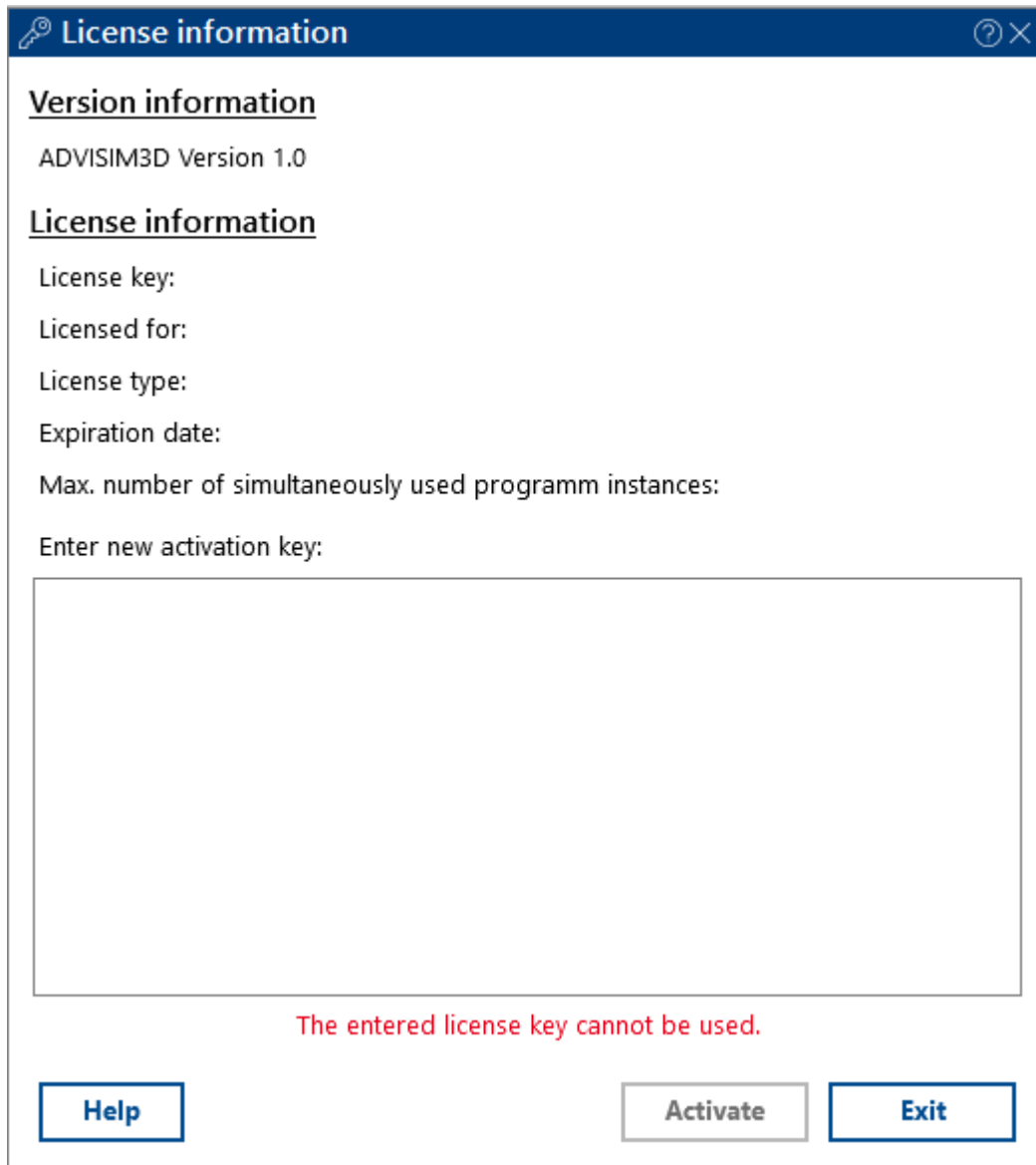
Enter new activation key:

No license could be found.

Help      Activate      Close

With "Apply" the entered license key is validated and used. If you close this dialog, ADVISIM<sup>3D</sup> is also closed.

If you enter an incorrect code, you will get this dialog:



The image shows a software dialog box titled "License information" with a key icon on the left and a help/cancel icon on the right. The dialog is divided into two sections. The first section, "Version information", displays "ADVISIM3D Version 1.0". The second section, "License information", contains labels for "License key:", "Licensed for:", "License type:", "Expiration date:", and "Max. number of simultaneously used programm instances:". Below these is a label "Enter new activation key:" followed by a large, empty rectangular text input field. At the bottom of the dialog, there is a red error message: "The entered license key cannot be used.". Three buttons are located at the bottom: "Help" on the left, "Activate" in the center, and "Exit" on the right.

**License information**

Version information

ADVISIM3D Version 1.0

License information

License key:

Licensed for:

License type:

Expiration date:




Max. number of simultaneously used programm instances:

Enter new activation key:

The entered license key cannot be used.

**Help** **Activate** **Exit**

If the license key is valid this dialog is presented:

 **License information**  

**Version information**  
ADVISIM3D version 2.0.0.10254

**License information**  
License key: 1234567890  
Licensed for: ADVITEC Informatik GmbH  
License type: Concurrent  
Expiration date: 31/12/2099  
Instances: ⓘ 1

Enter new activation key:

The license key entered is now in use.

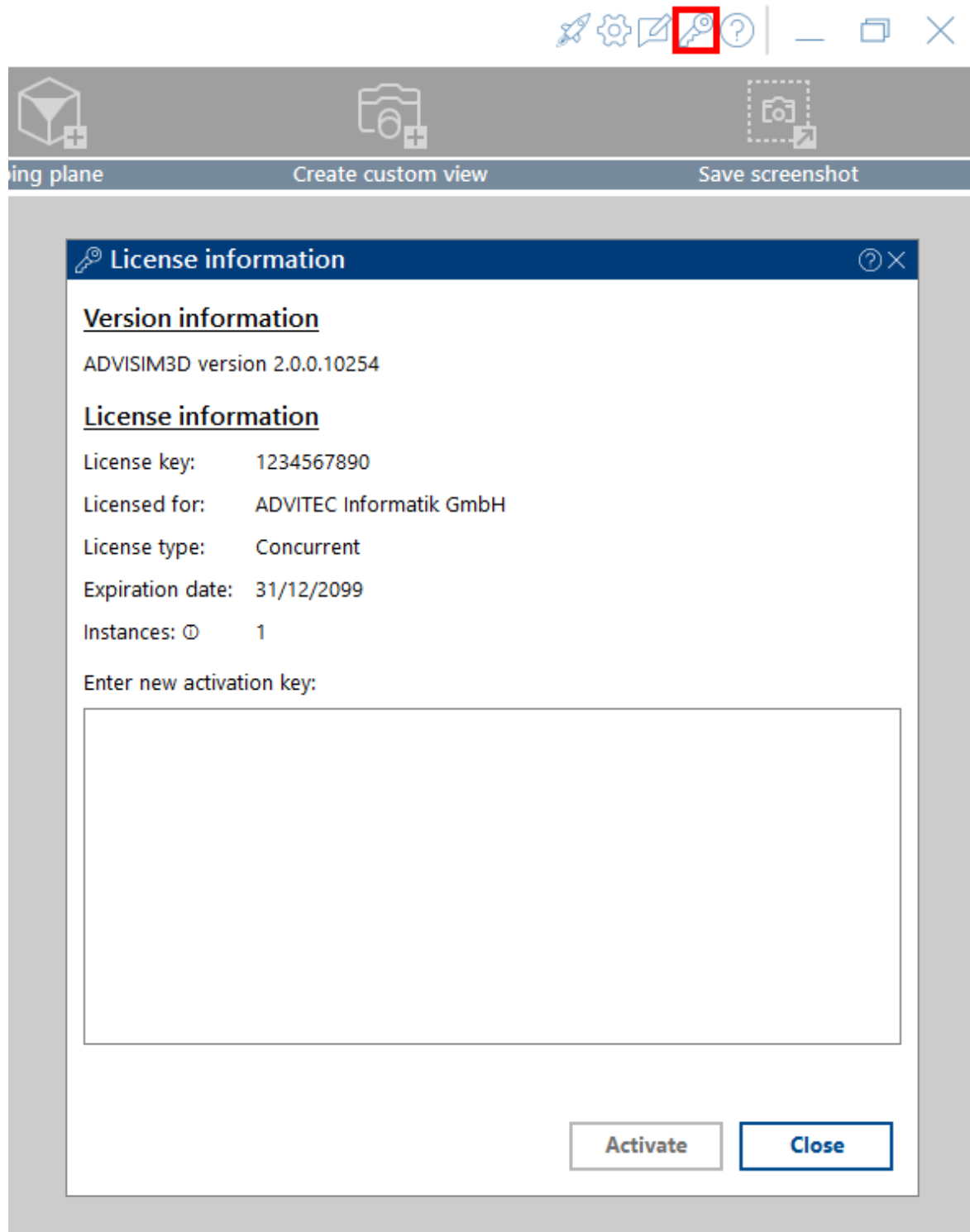
Activate

Close

If you have any problems please contact us via our [contact page](#).

## 4.20.2 Show license information

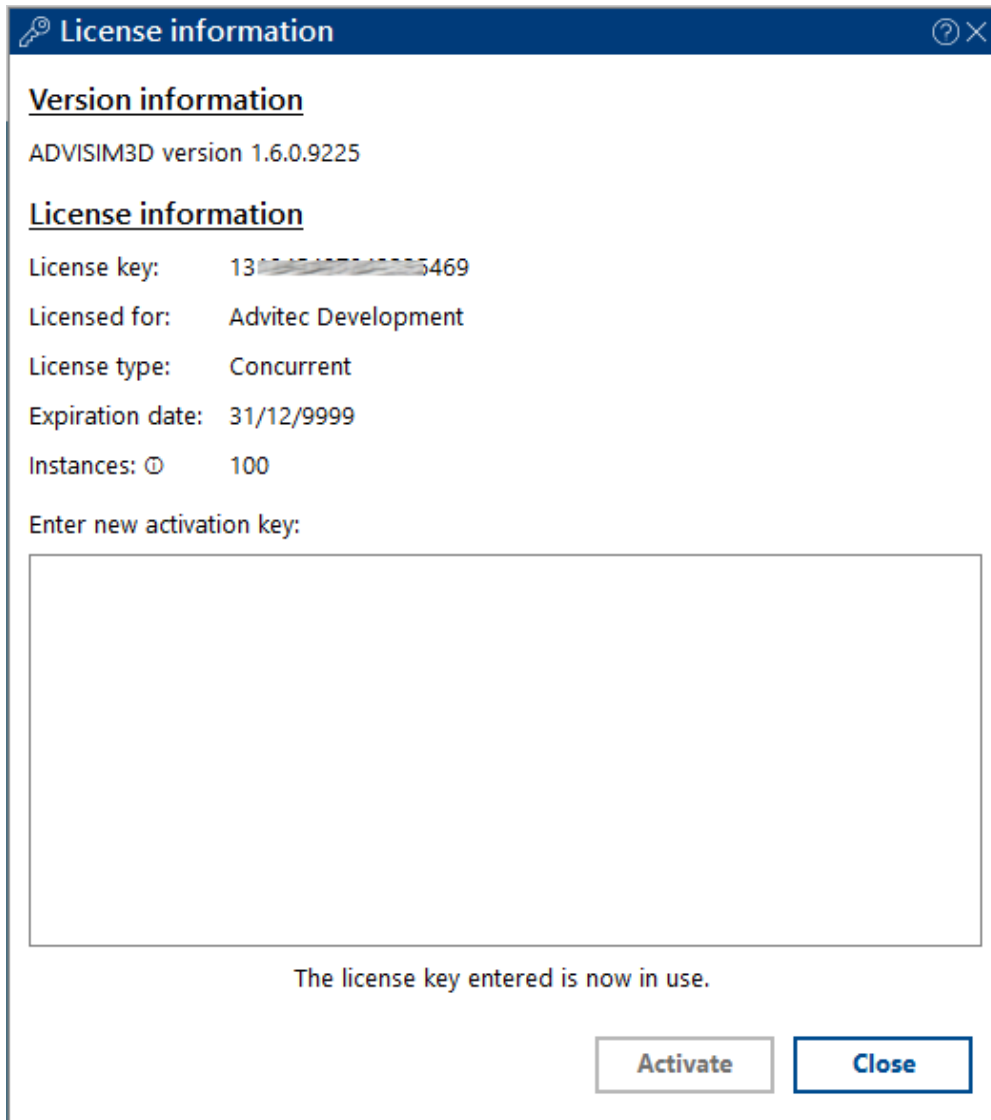
Pull up the dialog via the step shown.



Here you get all information about your license and can enter your license key.

Find out about ADVISIM<sup>3D</sup>'s license period and how many users may use ADVISIM<sup>3D</sup> simultaneously.

If you enter a new license key, the corresponding information about the license is displayed immediately.



The image shows a software dialog box titled "License information" with a key icon on the left and help and close icons on the right. It contains two sections: "Version information" and "License information". The "Version information" section displays "ADVISIM3D version 1.6.0.9225". The "License information" section lists details: "License key: 13[redacted]5469", "Licensed for: Advitec Development", "License type: Concurrent", "Expiration date: 31/12/9999", and "Instances: 100". Below this is a text input field labeled "Enter new activation key:". At the bottom, a message states "The license key entered is now in use." and there are "Activate" and "Close" buttons.

**License information**

Version information

ADVISIM3D version 1.6.0.9225

License information

License key: 13[redacted]5469

Licensed for: Advitec Development

License type: Concurrent

Expiration date: 31/12/9999

Instances: 100

Enter new activation key:

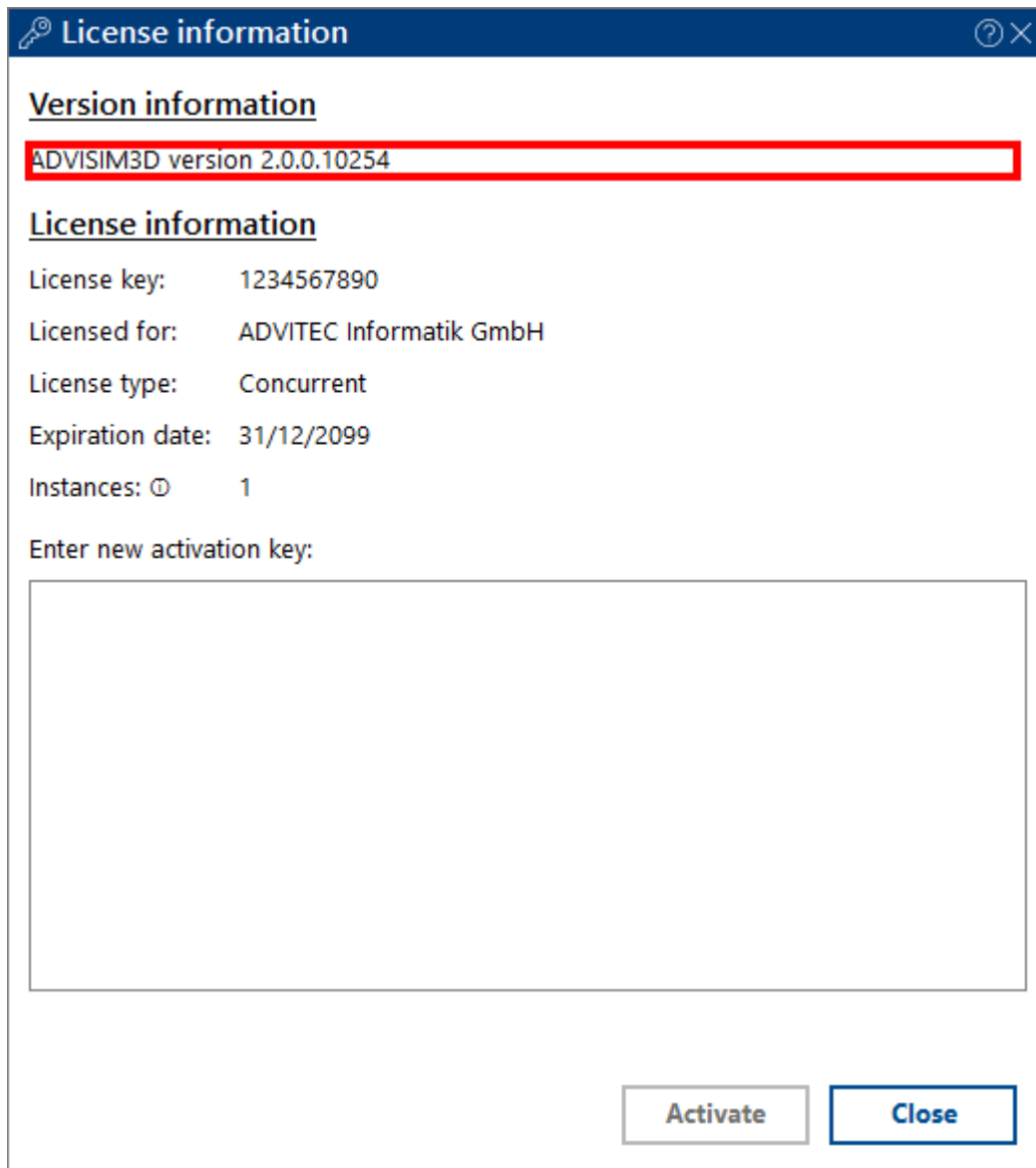
[Text input field]

The license key entered is now in use.

**Activate** **Close**

## ADVISIM<sup>3D</sup> Version

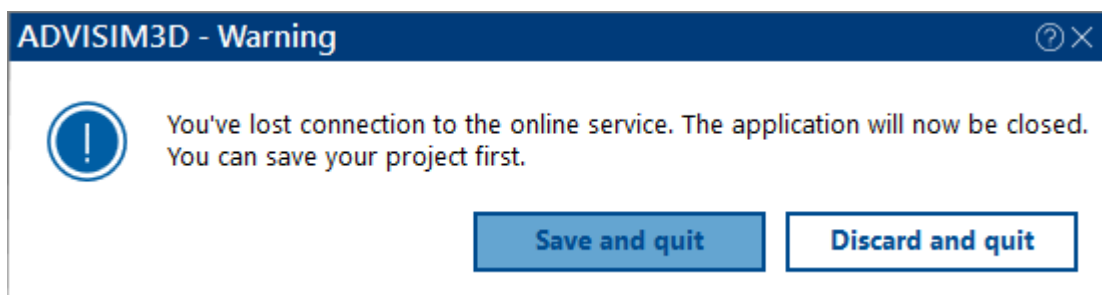
Below the license information you will find the current program version.



### 4.20.3 License service

ADVISIM<sup>3D</sup> cyclically checks the validity of the license information. This requires a permanent Internet connection.

If this connection fails, ADVISIM<sup>3D</sup> will close with the following error message.



You can still [save](#) your current project before exiting ADVISIM<sup>3D</sup>.

## V System documentation

Here you will learn how to put ADVISIM<sup>3D</sup> into operation.

We recommend using the latest drivers for your hardware as well as the latest version of Windows.

### 5.1 System requirements

For ADVISIM<sup>3D</sup> to work properly, the following system requirements must be met:

- 64-bit Windows operating system, Windows 10 or later
- Graphics card and graphics driver with OpenGL 4.0 or higher
- 32 GB RAM
- Internet connection
- CAD files in STEP format

Required and installed by ADVISIM<sup>3D</sup>

- Microsoft .NET-Framework 4.5
- Microsoft Visual C++ 2013 Redistributable x64



## VI FAQs

### Frequently Asked Questions (FAQs)

Here you will find a collection of useful answers to various questions and a list of known problems.

#### Content FAQ

- [Why is no spray pattern visible?](#)
- [Why does ADVISIM3D start with a Windows firewall query?](#)
- [Why am I getting a warning that my version is about to expire?](#)
- [Why am I getting unexpected simulation results?](#)

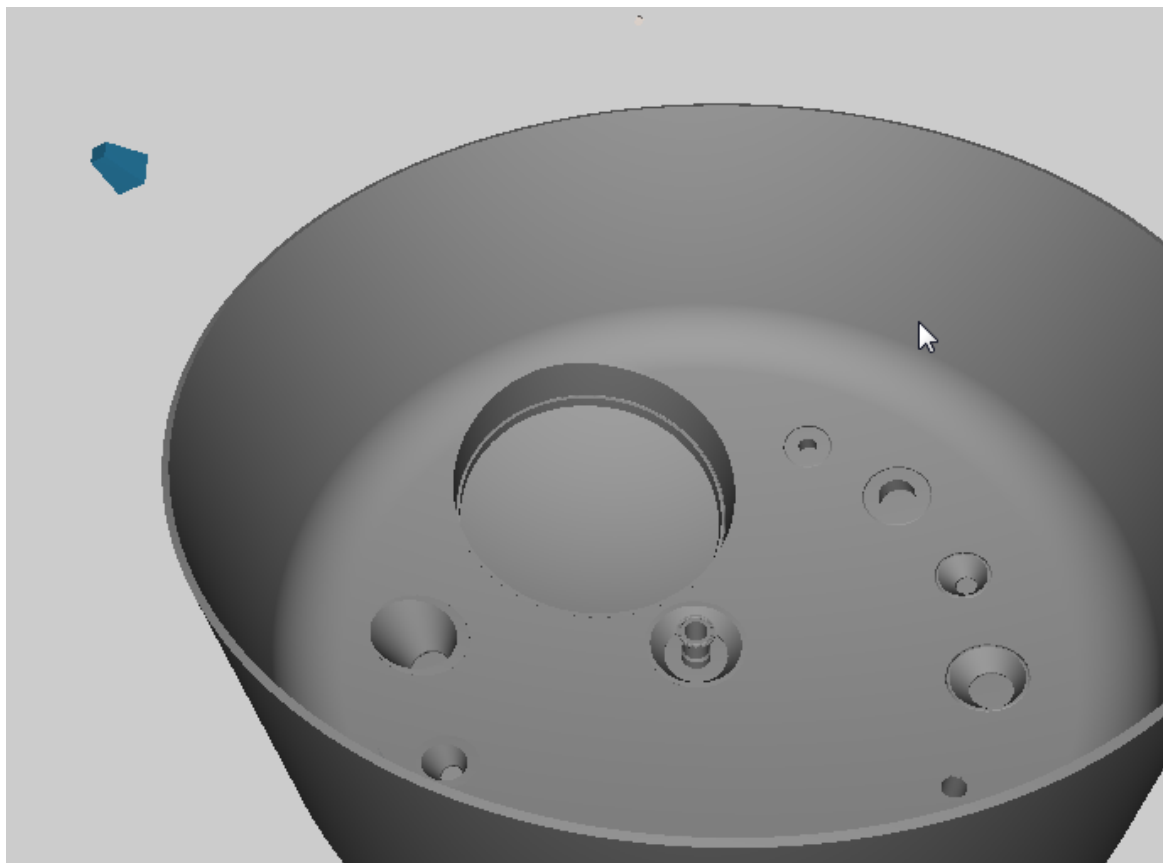
#### Known Problems

- [Transfer of simulation results](#)
- [Incorrect display of clipping planes](#)
- [Localization of nozzles generated from model part](#)
- [Pressure change of nozzles has no effect on workflow area](#)

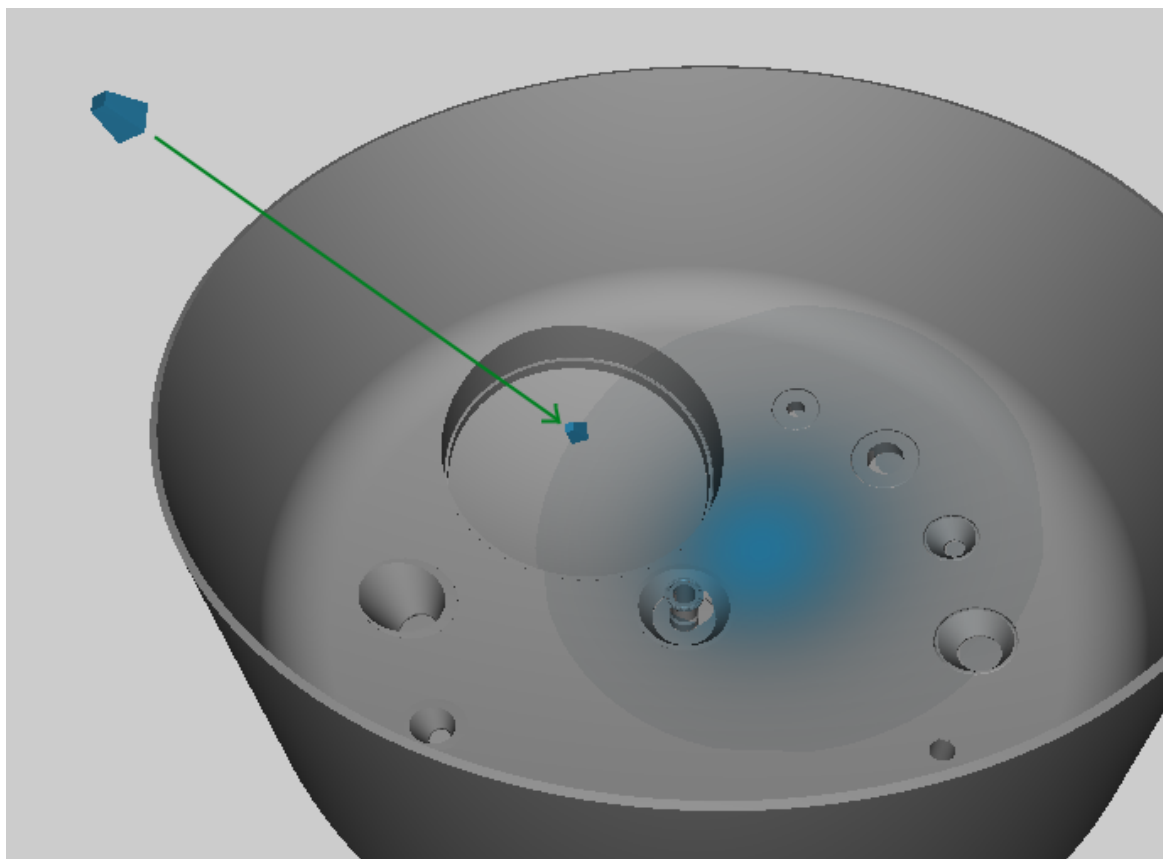
### 6.1 Spray pattern not visible

#### Why is no spray pattern visible?

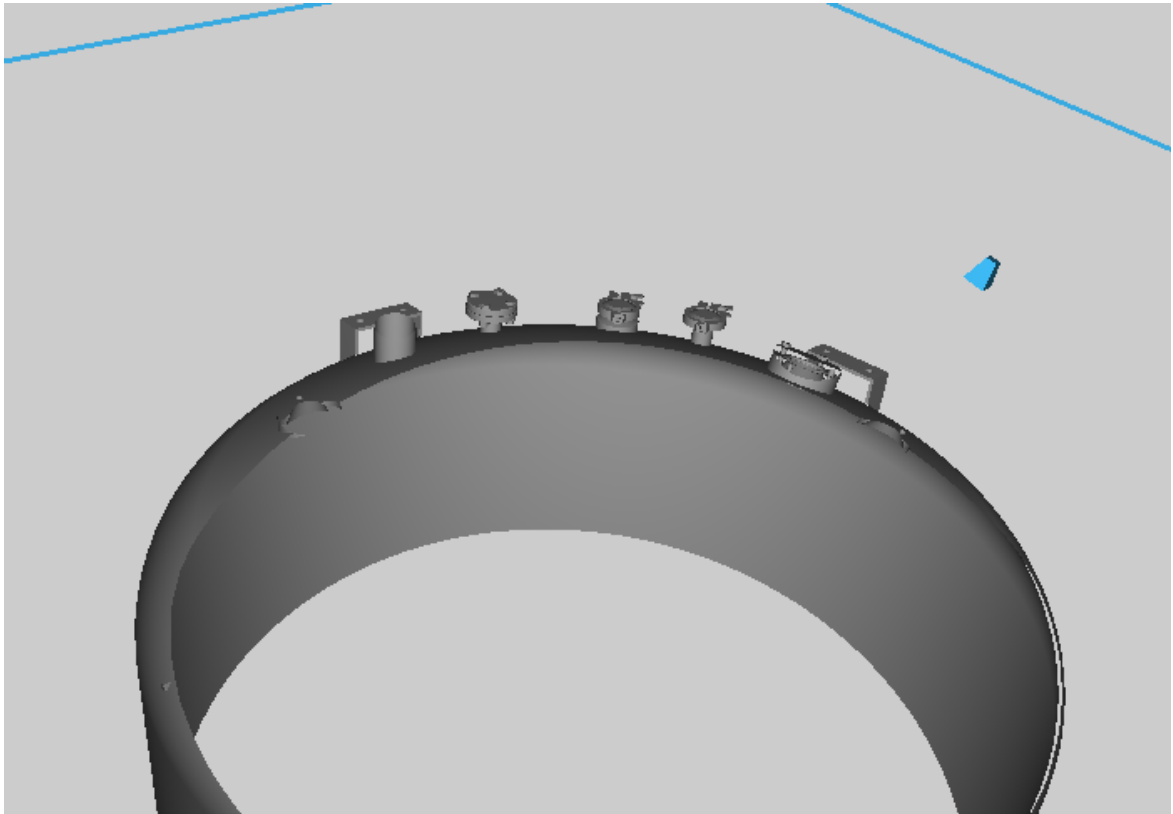
Since nozzles have a certain workflow area, it can happen that an inserted nozzle does not show a direct spray pattern. This happens if the nozzle is too far away from the surface to be cleaned.



[Move](#) or [relocate](#) the nozzle along its spray direction to see a spray pattern.



If the spray pattern lies on an area that is hidden by a clipping plane, this plane also remains invisible.



## 6.2 Firewall request at startup

### Why does ADVISIM<sup>3D</sup> start with a Windows firewall query?

Since ADVISIM<sup>3D</sup> accesses the online menu database, the program connects to this database at startup. This requires the permission of the firewall.

## 6.3 Version expired

### Why am I getting a warning that my version is about to expire?

A message of this type informs you that your current version will soon expire. You can no longer start ADVISIM<sup>3D</sup> after this date. By then you should update ADVISIM<sup>3D</sup> to the latest version.



After this expiration date you will receive the following message:



A new version is available at: [www.advisim3d.com/download](https://www.advisim3d.com/download)

## 6.4 Unexpected simulation results

### Why am I getting unexpected simulation results?

The simulation results depend on whether the graphics card interprets them correctly. If the results are very unrealistic, we ask you to update the driver of your graphics card to the latest version. If a driver update does not help, [contact us](#).

## 6.5 Known issues

Here is a list of known ADVISIM<sup>3D</sup> problems:

- [Transfer of simulation results](#)
- [Incorrect display of clipping planes](#)
- [Localization of nozzles generated from model part](#)
- [Pressure change of nozzles has no effect on workflow area](#)

### 6.5.1 Transfer of simulation results

#### Steps to reproduce

If you have run a simulation and saved the results, it is generally possible to copy the \*.smkr and the folder with the simulation data to another computer. If the STEP model is available (as described [here](#)), ADVISIM<sup>3D</sup> can load the stored data.

#### Problem

In this case it may happen that the results look different. The reason for this is that the generated grid can differ from computer to computer. Since the results are stored at the nodes of the grid, if the grid is different, the results may be stored at different locations.

### 6.5.2 Incorrect display of clipping planes

When [cut edges are displayed](#), it may happen that they are incorrect. It is possible that a model part was recognized as a cut edge that was not trimmed.

### 6.5.3 Localization of nozzle generated from model part

When [converting a model part into a nozzle](#), the nozzle is inserted with the position and orientation of the model part. If the position cannot be picked out, it is possible that ADVISIM<sup>3D</sup> inserts the nozzle in the coordinate origin.

### 6.5.4 Pressure change of nozzle has no effect on workflow area

The [working pressure can be defined](#) in the nozzle properties area. This has an effect on the volume flow, the spray pattern and, in the case of rotation nozzles, also on the rotational speed. However, the workflow area covered by the nozzle remains constant and represents the maximum workflow area.

## VII Index

### #

[3D coordinate system](#)

[3D toolbar](#)

### A

[Ablated layer thickness](#)

[Activate license](#)

### C

[Change clipping plane name](#)

[Change clipping plane normal](#)

[Change clipping plane position](#)

[Change constraint distance](#)

[Change constraint orientation](#)

[Change custom camera view name](#)

[Change nozzle direction](#)

[Change nozzle distance](#)

[Change nozzle name](#)

[Change nozzle position](#)

[Change nozzle type](#)

[Change nozzle up-vector](#)

[Change view mode](#)

[Change working pressure](#)

[Choose 3D views for report](#)

[Clipping plane angle to local reference planes](#)

[Clipping plane interaction](#)

[Clipping plane properties area](#)

[Color scale area](#)

[Configure simulation](#)

[Constraint add along a model part axis](#)

[Constraint add parallel to a model part plane](#)

[Constraint along a model part axis](#)

[Constraint interaction](#)

[Constraint parallel to a model part plane](#)

[Constraints properties area](#)

[Constraint types](#)

[Contact](#)

[Convert model part to nozzle](#)

[Coordinate system planes](#)

[Create new project](#)

[Custom camera views](#)

[Custom view properties area](#)

## D

[Delete clipping plane](#)

[Delete constraint](#)

[Delete custom camera view](#)

[Delete nozzle](#)

[Direct volume application including footprint](#)

## E

[Error message for invalid constraint combination](#)

[Exit simulation](#)

[Export nozzle](#)

## F

[FAQ](#)

[File import](#)

[Firewall request at startup](#)

[First steps](#)

## G

[General clipping plane properties](#)

[General constraint properties](#)

[General nozzles properties](#)

## H

[Hide clipping planes](#)

[Hide model parts](#)

[Hide nozzle](#)

## I

[Insert clipping plane](#)

[Insert constraint](#)

[Insert custom camera views](#)

[Insert nozzle](#)

[Invert clipping plane normal](#)

[Invert nozzle spray direction](#)

## K

[Keyboard interaction](#)

[Known issues](#)

## L

[License management](#)

[Linear movement](#)

## M

[Mark clipping planes](#)

[Model part interaction](#)

[Mouse interaction](#)

[Move clipping plane along its orientation](#)

[Move nozzle](#)

[Move nozzle along its spray direction](#)

[Move nozzle with constant target point](#)

## N

[Nozzle category](#)

[Nozzle interaction](#)

[Nozzle selection dialog](#)

## O

[Open context menu](#)

[Open project](#)

[Open simulation](#)

## P

[Playback mode](#)

[Pipette](#)

[Print report](#)

[Print report general information](#)



[Print report nozzle information](#)  
[Print report non-visible model parts](#)  
[Print report page settings](#)  
[Program structure](#)  
[Projects](#)  
[Properties area](#)  
[Properties area custom views](#)  
[Properties area for nozzles](#)

## Q

[Quick navigation](#)  
[Quick Start Guide](#)

## R

[Redo actions](#)  
[Release Notes](#)  
[Reset nozzle direction](#)

## S

[Save project](#)  
[Save simulation](#)  
[Set movement](#)  
[Set visibility of all scene elements](#)  
[Settings](#)  
[Select clipping plane](#)  
[Select nozzle](#)  
[Show clipping planes](#)  
[Show constraint name](#)  
[Show license information](#)  
[Show nozzle](#)  
[Show model part](#)  
[Show movement line](#)  
[Show workflow area](#)  
[Simulation](#)  
[Simulation views](#)  
[Simulation properties area](#)  
[Simulation result view](#)  
[Simulation modes](#)  
[Spray pattern not visible](#)  
[Spray balls](#)

- [Standard views](#)
- [Start and continue simulation](#)
- [Stop simulation](#)
- [Structuretree](#)
- [System documentation](#)
- [System requirements](#)

## T

- [Take a screenshot](#)
- [Target stream cleaner](#)
- [Tilt clipping plane](#)
- [Time navigation](#)
- [Top direction of the model](#)
- [Turn nozzle around its spray direction](#)

## U

- [Undo actions](#)
- [Undo / Redo actions](#)
- [Unexpected simulation results](#)

## V

- [Version expired](#)
- [View control](#)