CADMOULD & VARIMOS
by SIMCON

WHAT'S NEW IN VERSION 17.1.1

Display of runner geometries during creation - Reasons for shrinkage and warpage - New API functions

Do you have any questions?
Please click here or scan the QR code to reach our support team

What's new in CADMOULD V17.1.1

Realistic representation of runner segments during runner build-up

The design of a suitable runner system is one of the key tasks/challenges when designing injection molds. If the dimensions and geometries of the runner are not matched to the application, it leads to issues during filling, e.g. early sealing or to high stresses. The packing pressure supply can also be negatively affected. To simulate runner systems in CADMOULD, users have two options: Create so-called segments in CADMOULD or import them as a CAD model.

With the aid of **CADMOULD segments** users can create runner systems directly in CADMOULD. So far, the segments have been represented as cylinders whose diameter corresponds to the hydraulic diameter of the actual geometry. This meant that the runners were always displayed in a simplified form. It was not possible to display the exact geometry as this information was lost in the representation.

CADMOULD 17.1 now makes it possible to display the **actual runner geometry**. The new display of the runner system provides a realistic overall impression and allows users to check the plausibility of the entries quickly and easily.

The option to change the view is in the **runner segments** dialog above the different cross-sections (see **Figure 1**). The **3D shape** function can be used to display the runner geometry during runner build-up. By default, only the **selected segment** is initially displayed in the actual geometry. The **Show all** function activates the view of all segments already created and thus provides an overall impression of the system. The degree of transparency of the display can be adjusted using the **3D Shape Transparency** function.

The field for aligning individual segments (**Rotation [°]**) is also new. The rotation angle entered there is used for the optical alignment of the runner segments, e.g. in ejection direction. This setting is saved in the project. This function is for display purposes only and does not affect the simulation result.

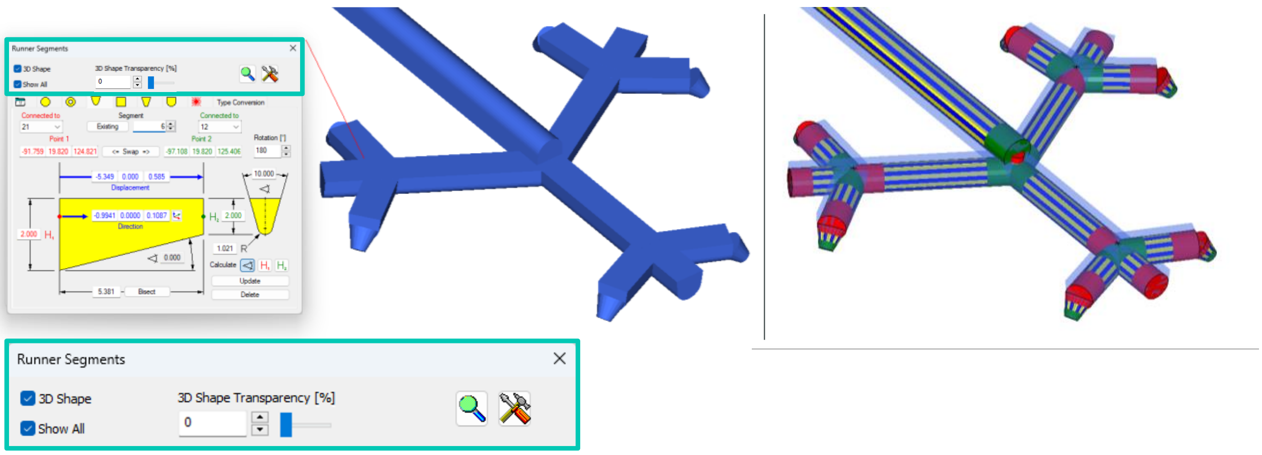

***Figure 1**: Runner segments dialog with new display options.*

Expansion of the control options for cascadic injection molding

The opening time of a nozzle in cascade injection molding cannot always be defined by the filling time of a node or as an absolute time. Particularly if nozzles are to open relative to each other, the explicit opening time of the first nozzle must be known to be able to enter that of the next nozzle. Previously it was necessary to determine the time of overflow manually using a preliminary simulation and then implement this in the second simulation. Especially if the filling profile, time, and speed have not yet been determined, this approach causes frustration and costs valuable time. In CADMOULD V17.1, the valve gate nozzle control options have now been extended to include a **time delay.** This can be found in the dialog under the control type **CP reached + [s]** (see **Figure 2**). Sensors are used as control points, which are placed close to the nozzle, for example. The time between the first contact of the sensor with the flow front and the opening of the nozzle is entered as a parameter in seconds [s]. This extension supports the user in determining a suitable cascade profile in a shorter time and transferring it to the application.

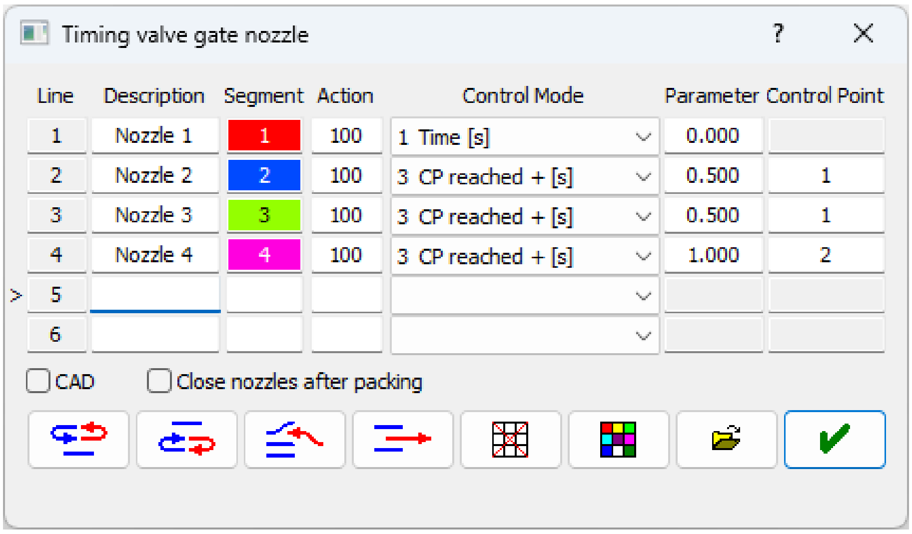

Figure 2*: valve gate control dialog with new control option*

Air traps results optimized and extended

Air traps can significantly influence the quality of the molded part and lead to burn marks, for example. The position and extent of the air traps are significantly influenced by the process control and the gate location. Consequently, the detection of critical air traps is not trivial.

CADMOULD already offers the option of detecting air traps and shows you areas where venting is required. This is supplemented by **two new features** in CADMOULD 17.1:

The **detection of air traps** has been **optimized** and now uses **more precise criteria** to identify them. This means that fewer locations are marked as air traps by filtering out non-critical areas. This makes it **easier** for the user to identify the **relevant locations** and assess whether a venting is necessary.

The following filling pattern (**Figure 3**) shows a typical air trap where venting is required.

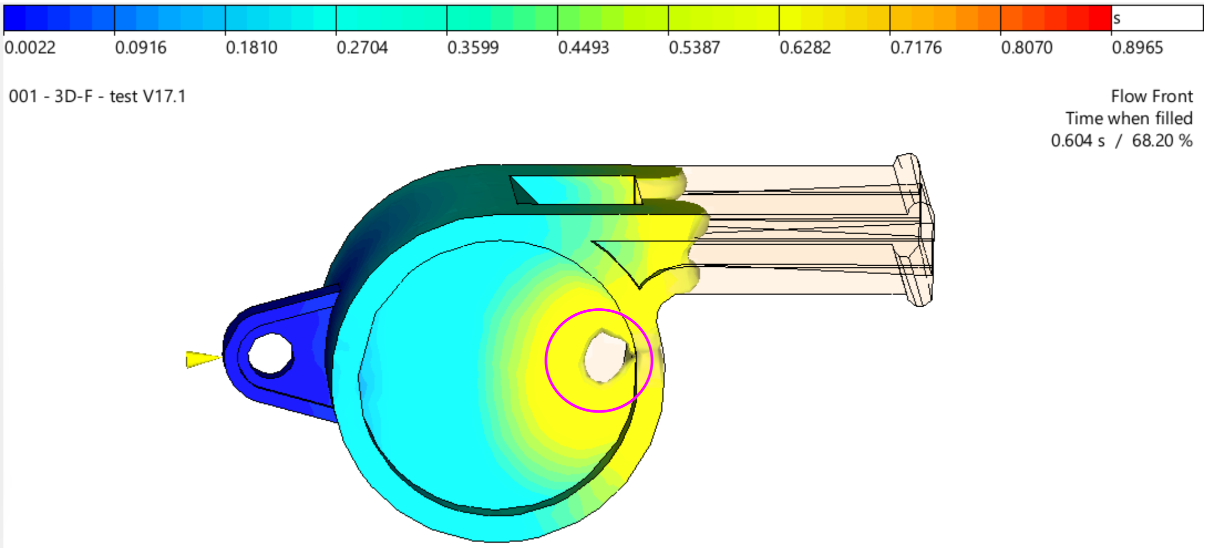

Figure 3*: Snapshot of the molded part filling with air entrapment.*

To display the air traps, activate **Air traps**. You will find the corresponding button in the **result selection** (**Figure 4**).

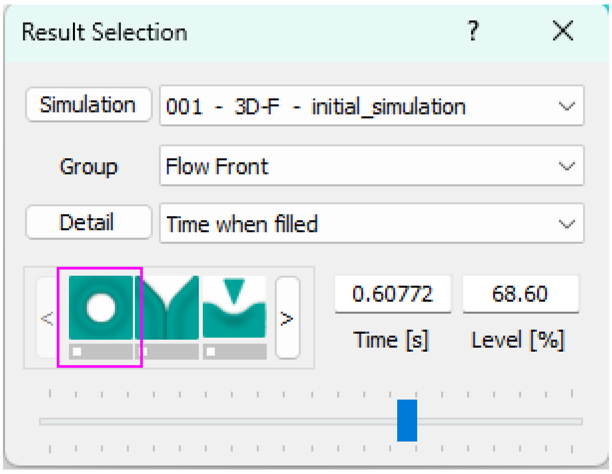

Figure 4*: Result selection - air traps*

Figure 5 shows the detected air trap positions (white spherical elements), including the part area mentioned above. The other air entrapment positions are in the area of merging flow fronts and at the flow paths ends.

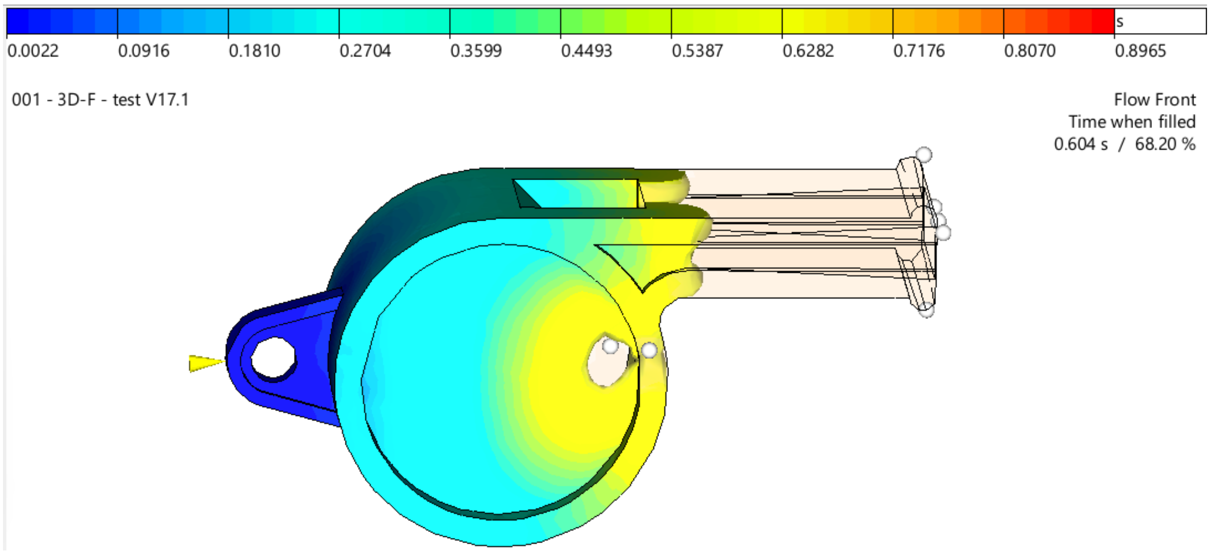

Figure 5*: Detected air traps*

The assessment of whether a venting should be included must still be made by the user, as CADMOULD does not yet have any knowledge of the parting plane in the tool. However, there is further support for this with the new **Compression spots** function.

You can find this result in the **result selection** under **Flow front - Compression spots**. It shows the pressure that would occur in the various areas if there were no venting (see **Figure 6**). The scale is automatically limited to 500 bar. The value 500 bar is a predefined limit which usually highlights critical areas well. However, it does not indicate whether an air entrapment is critical or not. Look for local maxima. These are the places where venting is necessary. It may be necessary to lower the upper limit of the scale further to better highlight local differences.

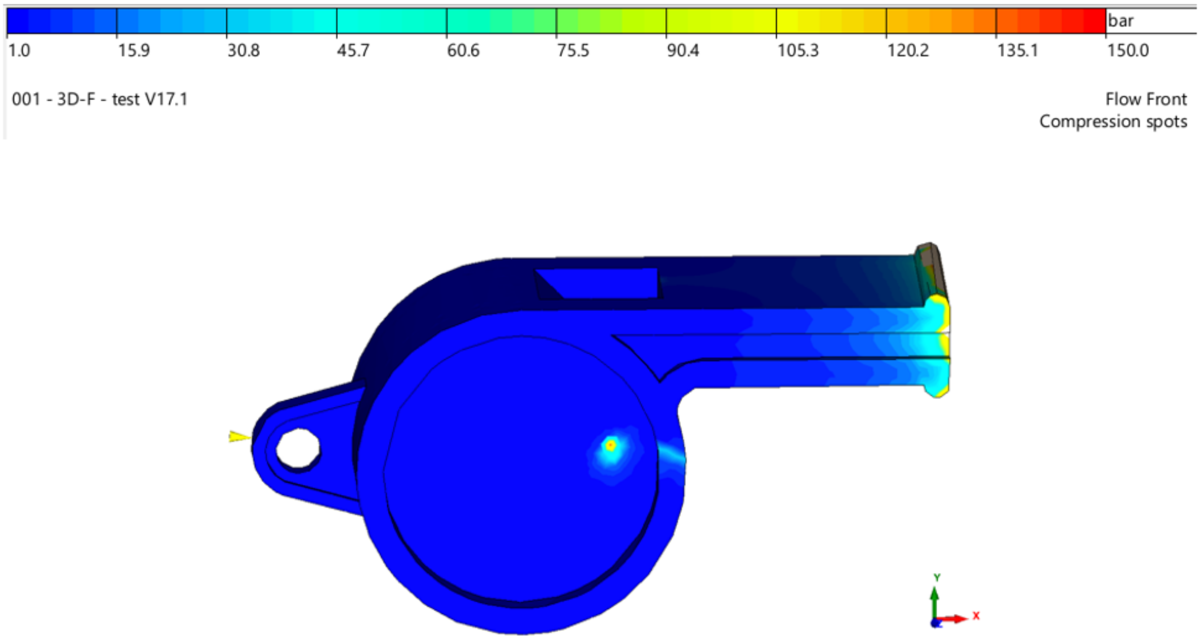

***Figure 6**: New result: Compression points*

Weld line results optimized and extended

Weld lines have a significant influence on the quality and strength of molded parts, and it is not trivial to predict their exact position and properties. CADMOULD already offers the **weld line result** to support you in recognizing such areas. The algorithms of the **Reduce artefacts** function introduced in CADMOULD 17.0 have been further optimized to highlight critical areas even better.

In addition, diagrams with results relevant to the weld line quality can now be called up:

- Temperature / time
- Pressure / time
- Velocity / time

Use the **drop-down menu** in the **weld line dialog** to select the desired result and click on the weld line while holding down the control key (see **Figure 7**).

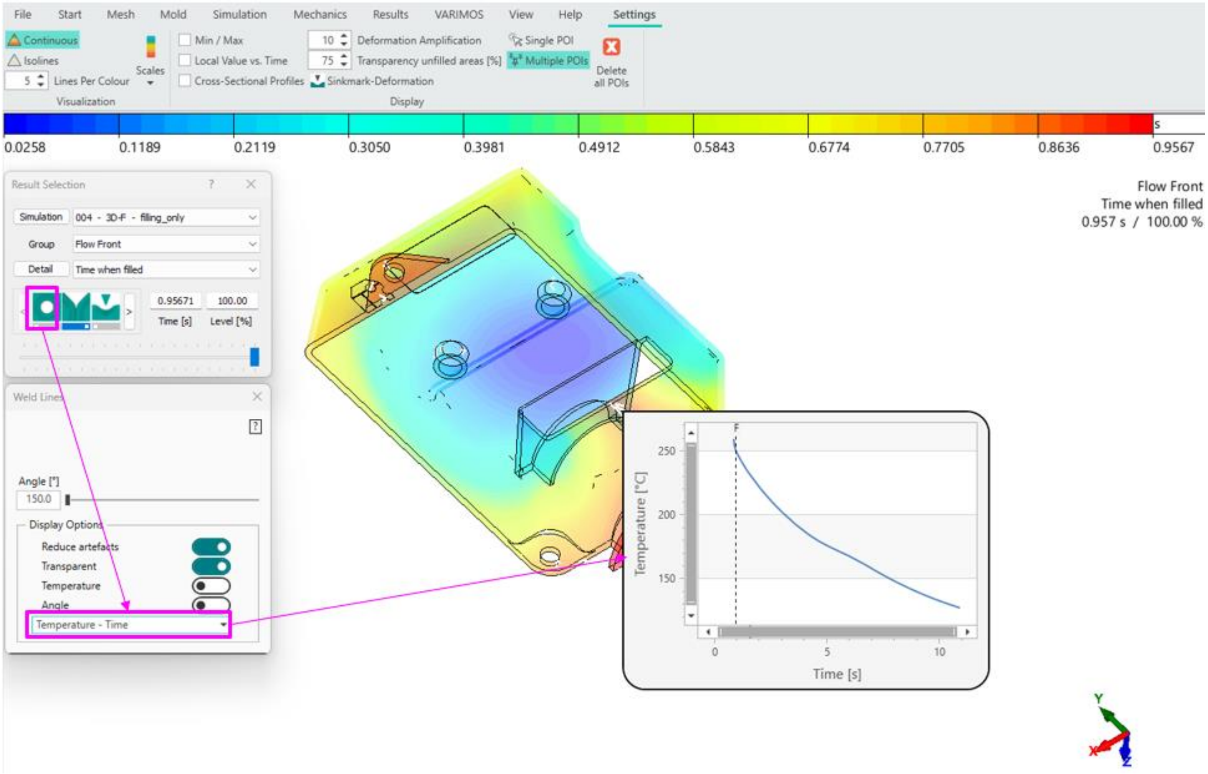

***Figure 7**: Seam dialog: Diagrams (example: temperature/time)*

A diagram appears in which the selected result is plotted over time. The **multiple POIs** option is automatically activated. Otherwise, the diagram would disappear immediately.

Optimized display of sink marks

Sink marks are a typical problem in the injection molding of plastics. They ruin the aesthetic impression of housings or other visible components.

For this reason, we have completely revised our sink marks analysis in CADMOULD and divided it into two sections. This new approach makes it possible not only to quantify the effects of sink marks, but also to make them visually available. The details of this approach are described below:

New display options: Clarity and precision

The **sink marks** result in the **part quality** group previously represented a traffic light indicator for sink marks. This allowed critical areas to be identified and then quantified using the **thickness shrinkage** result. This approach has now been **significantly simplified**: The revised result highlights all areas that have a sink mark in color (false color display). The sink mark depth can be read directly from the color scale (see **Figure 8**).

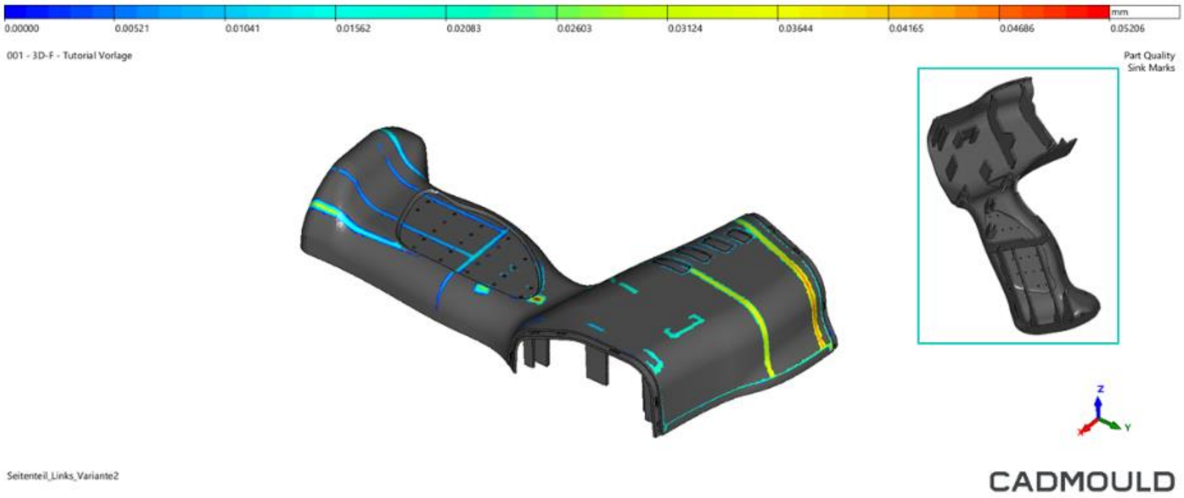

***Figure 8**: Representation of the sink mark depth in CADMOULD V17.1*

Realistic preview - One click for the reality check

In addition to the **display** shown above, there is a new view that enables a realistic preview. You can switch between the display options in the **Settings** ribbon by clicking on the **Display sink marks** button (see **Figure 9**).

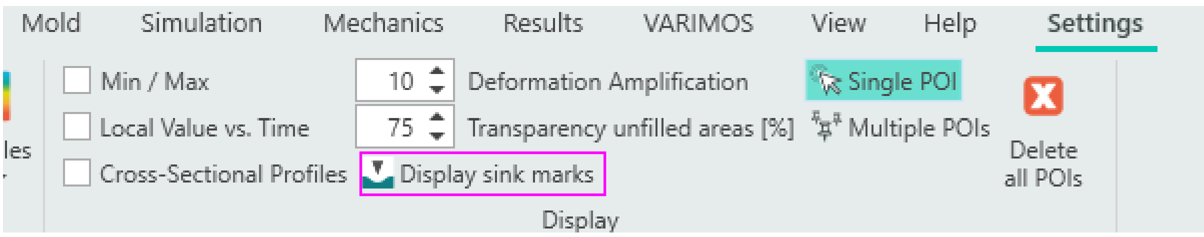

***Figure 9**: Display option for sink marks: "Display sink marks"*

Press the button again to switch back to the quantitative display.

This option allows the user to visualize the actual effect of the calculated defect (see **Figure 10**).

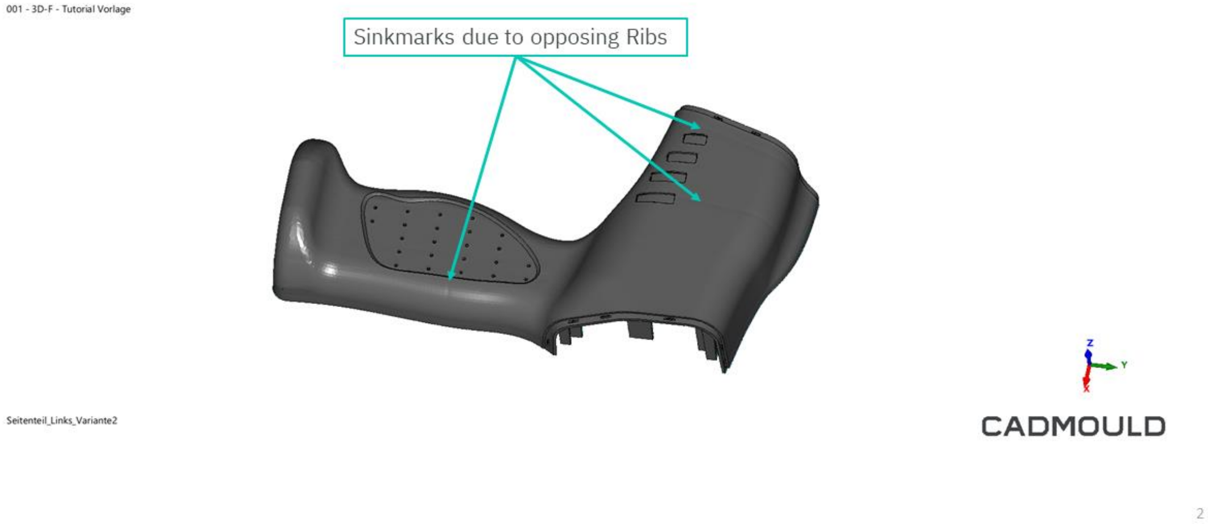

***Figure 10**: Illustration of the sink mark deformation*

For the most authentic representation, you can select a color for your component in the **Geometry Explorer** that corresponds to that of your material. The gloss level can be set under **View - Options - Other - Lighting**.

Improved shrinkage and warpage calculation for certain geometries

CADMOULD 17.1 offers several algorithmic optimizations that improve the accuracy of the simulation. To this end, the system continuously compares calculation results with real measurement results to calibrate and expand the underlying models.

One of the most important improvements in this version is the **optimized calculation of shrinkage and warpage**:
The optimization of the shrinkage and warpage calculation concerns anisotropic materials, **especially fiber-reinforced materials, in the area of branches (e.g. rib feet)**. In the past, excessive thickness shrinkage and warpage were predicted in these areas in certain situations. The adjustments make these areas more realistic, and investigations on rib structures have shown the following:
The **thickness shrinkage and warpage** that occurs at the branches **tends to** be **reduced**.

Compared to earlier versions, the difference is greater on ribs with a transverse flow than on ribs with a longitudinal flow.

With **higher anisotropy,** the difference can become **greater** compared to earlier versions.

Important note about meshing:

Our recommendation is to **halve the suggested element edge length for the entire molded part at most.** If necessary (film gates, sink marks), finer meshing should only be carried out locally at these points. With this strategy, good results can be expected with fast calculation.

In the case of **very fine meshes,** the influence of the described optimizations may be reduced.

Please use your engineering expertise to check whether your empirical values regarding shrinkage and warpage results can be transferred to the latest CADMOULD version.

Influence analysis of shrinkage and warp-age (decomposition)

The reasons for component deformation (shrinkage and warpage) are complex. Local influences of the **cooling system,** but also the influence of **glass fibers** is among the main reasons for **locally different shrinkage potentials** and consequently **warpage**. Due to the complexity of the injection molding process, a clear differentiation and identification of the main effects is not trivial.

In addition to the already available option to simulate shrinkage and warpage and derive optimization options from this, CADMOULD 17.1 provides you with even **deeper insights into the effects**: In addition to the overall result, you can now evaluate the **influences** of local **fiber orientation** and **cooling.**

The activation and result display of the new feature is described below:

First, the **Shrinkage and warpage decomposition** function must be activated in the **Shrinkage and warpage** section of the **simulation options** and then the simulation must be started. The available calculation options are displayed below the function call (see **Figure 11**).

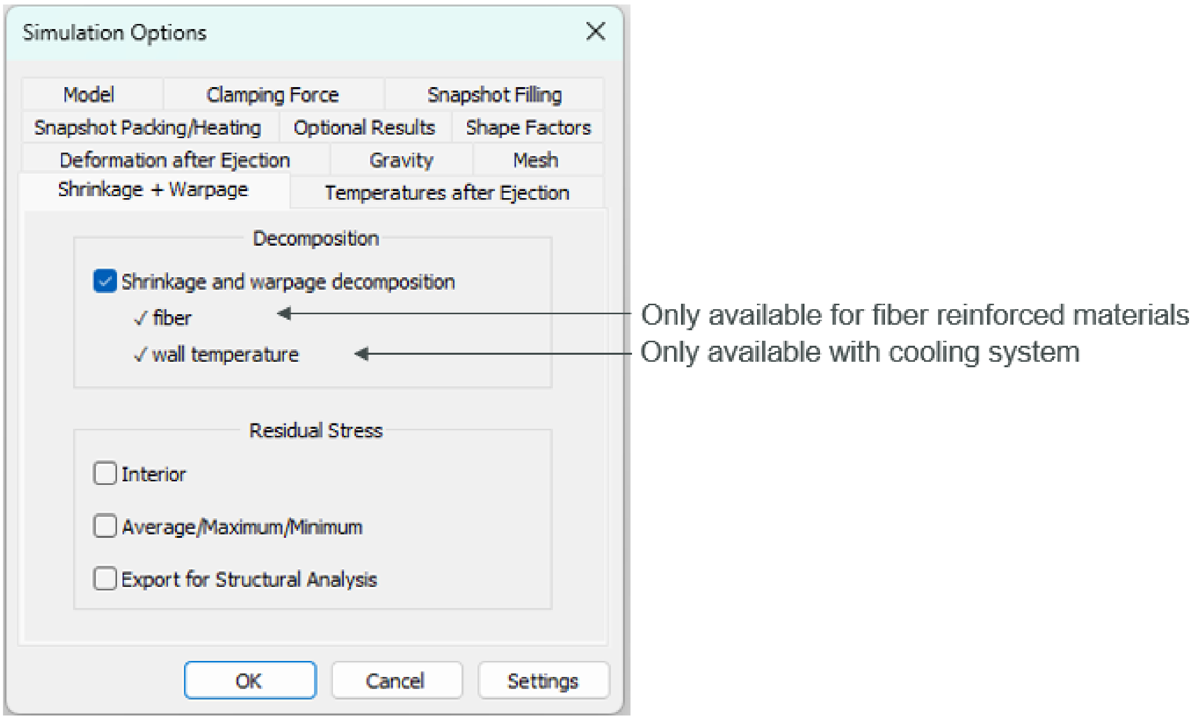

Figure 11*: Activation of S+W decomposition and available factors*

As more extensive calculations are required for this, this function is optional and not active by default (longer calculation times possible).

Once the simulation has been successfully calculated, new result groups are available:

- **S+W Decomposition: Fiber**
- **S+W Decomposition: Wall temperature**

Please note that the respective results are only available if you have calculated with cooling and/or a fiber-reinforced material.

The corresponding groups contain the same result details as the already known shrinkage and warpage result group:

- **X Deformation**
- **Y Deformation**
- **Z Deformation**
- **Deformation**
- **Shrinkage**
- **X Warpage**
- **Y Warpage**
- **Z Warpage**
- **Warpage**
- **Curvature Change**

However, the corresponding results represent the **local effect** resulting from the cooling or fiber influence. For the fiber influence, the results show the **difference** to a **random fiber distribution** (isotropic). In the case of the cooling system influence, these are the **differences** to a **uniform surface temperature** (assumption: opt. wall temperature from material database). **Figure 12** shows the result **S+W decomp. fiber - deformation**. The fiber structure results in a maximum deformation of about 0.3 mm in the corner areas of the molded part.

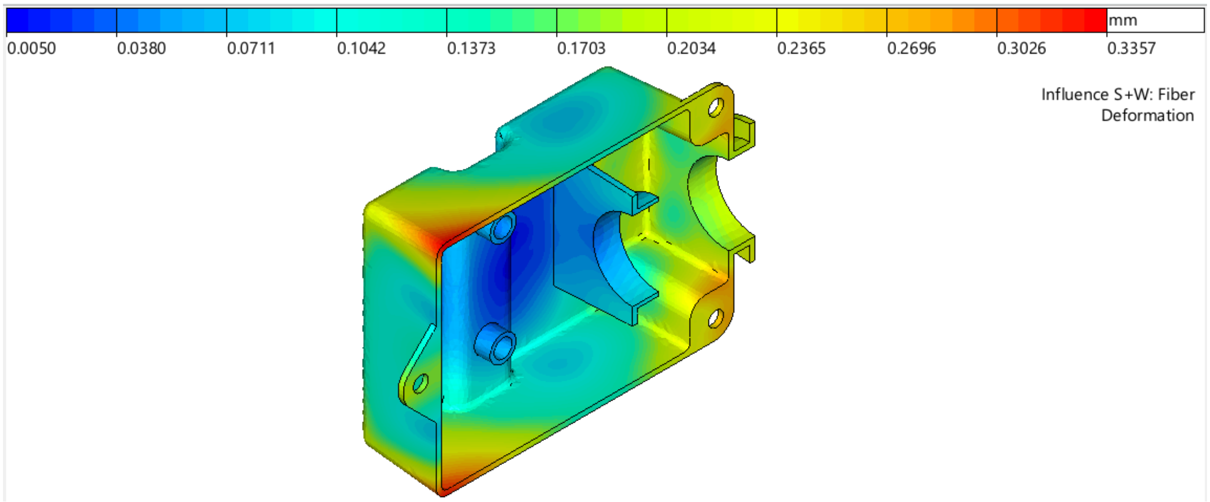

Figure 12*: S+W Decomposition: Fiber – Deformation*

Figure 13 shows the same result, but for the wall temperature effect.

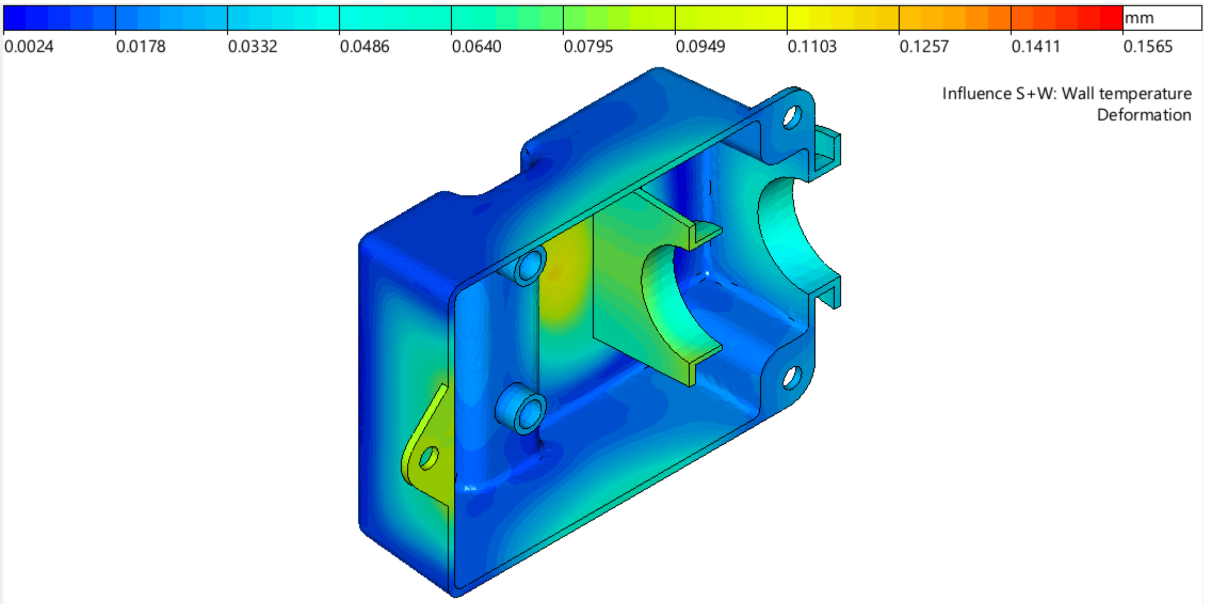

Figure 13*: S+W decomposition: wall temperature - deformation*

Here, the maximum deformations that can be attributed to the cooling system can be seen elsewhere. The effects are also significantly lower.

When interpreting the results, the absolute result should always be considered in relation to the respective effect to be able to assess the extent or significance.

The **deformed part** function can also be activated for a better assessment of the deformation direction. Furthermore, in the case of a fiber-reinforced material, the results can be overlaid with the **orientation function** to take the **local fiber orientation** directly into account in the result evaluation.

With the help of the additional results, you can now see what the deformations are due to and whether influences may compensate for each other.

You will also find the **S+W Decomp.** button in the **results ribbon**. After you have pressed the button, a new window opens with a tabular overview (see **Figure 14**).

Shrinkage and Warpage - Decomposition Overview

Copy

	Baseline				Isotrop				Unified Wall Temperature			
Name	Min	Max	Avg	Delta	Min	Max	Avg	Delta	Min	Max	Avg	Delta
X Deformation [mm]	-0.3483	0.3471	0.0000	0.6954	-0.4854	0.4858	0.0000	0.9712	-0.2996	0.2997	0.0000	0.5993
Y Deformation [mm]	-0.6908	0.7721	0.0000	1.4629	-0.7896	0.8626	0.0000	1.6522	-0.6050	0.6836	0.0000	1.2886
Z Deformation [mm]	-0.2378	0.3567	0.0000	0.5944	-0.2491	0.4969	0.0000	0.7459	-0.2256	0.3001	0.0000	0.5257
Deformation [mm]	0.0139	0.7765	0.2799	0.7626	0.0146	0.8638	0.3943	0.8492	0.0251	0.6875	0.2710	0.6624
Shrinkage [%]	0.1146	1.0654	0.4682	0.9508	0.2281	1.6117	0.7607	1.3836	0.1145	1.0409	0.4629	0.9264
X Warpage [mm]	-0.1726	0.1543	0.0000	0.3269	-0.1780	0.1766	0.0000	0.3546	-0.1523	0.1383	0.0000	0.2906
Y Warpage [mm]	-0.4289	0.4666	0.0000	0.8955	-0.2878	0.3653	0.0000	0.6531	-0.3454	0.3792	0.0000	0.7246
Z Warpage [mm]	-0.1467	0.3017	0.0000	0.4484	-0.1511	0.4113	0.0000	0.5625	-0.1730	0.2445	0.0000	0.4175
Warpage [mm]	0.0031	0.4736	0.1214	0.4704	0.0115	0.4136	0.1071	0.4021	0.0055	0.3845	0.1184	0.3790
Curvature Change [...	-0.7705	0.7901	0.0002	1.5606	-0.8506	0.7514	-0.0004	1.6020	-0.8131	0.8259	0.0001	1.6390

Results actual simulation

Isotropic Material behavior
(random fiber distribution)

Uniform Wall temperature

***Figure 14**: Shrinkage and Warpage – Decomposition Overview*

In addition to the results overview of the current simulation (baseline), the table shows the predicted results for a random fiber distribution (isotropic) and assuming a uniform wall temperature. The following results are available:

- **Min**: Minimum value
- **Max**: Maximum value
- **Avg**: Arithmetic mean value (Average)
- **Delta**: Span from min. to max.

This table can be used to **qualitatively/globally estimate** the range in which the values would be if **there were** a **random fiber distribution** or if the **wall temperature would be uniform**. The content of the table can be **copied** using the **copy function** at the top left of the window to process the data externally.

New 2K results

Realistic insert integration plays an important role in injection molding simulation. CADMOULD allows the precise calculation of thermal processes in the molded part and in the mold. For this purpose, the **temperature distribution** with all details such as sliders, ejectors, cooling channels and existing inserts can be considered. The analysis of the **thermal and mechanical interaction** of the insert with the component is extremely important, e.g. for analyzing shrinkage and warpage. The combination of thermal and mechanical influences during filling leads to **deformation** or **damage** to the **insert** and the **component** if the **process settings are incorrect**.

The integration of inserts is already possible in CADMOULD. **New result types** have now been implemented in CADMOULD 17.1 for **quick and easy analysis** of the **interactions** of **inserts** with **components** and **mold components.**

In addition to the already known functionalities of the **2K** results group, the **details Max. Surface temperature** and **Max. Shear stress** are available now:

The **max. surface temperature** describes the **maximum temperature** over **time** that is reached locally during filling and packing for each point on the surface of the inserts (see **Figure 15**).

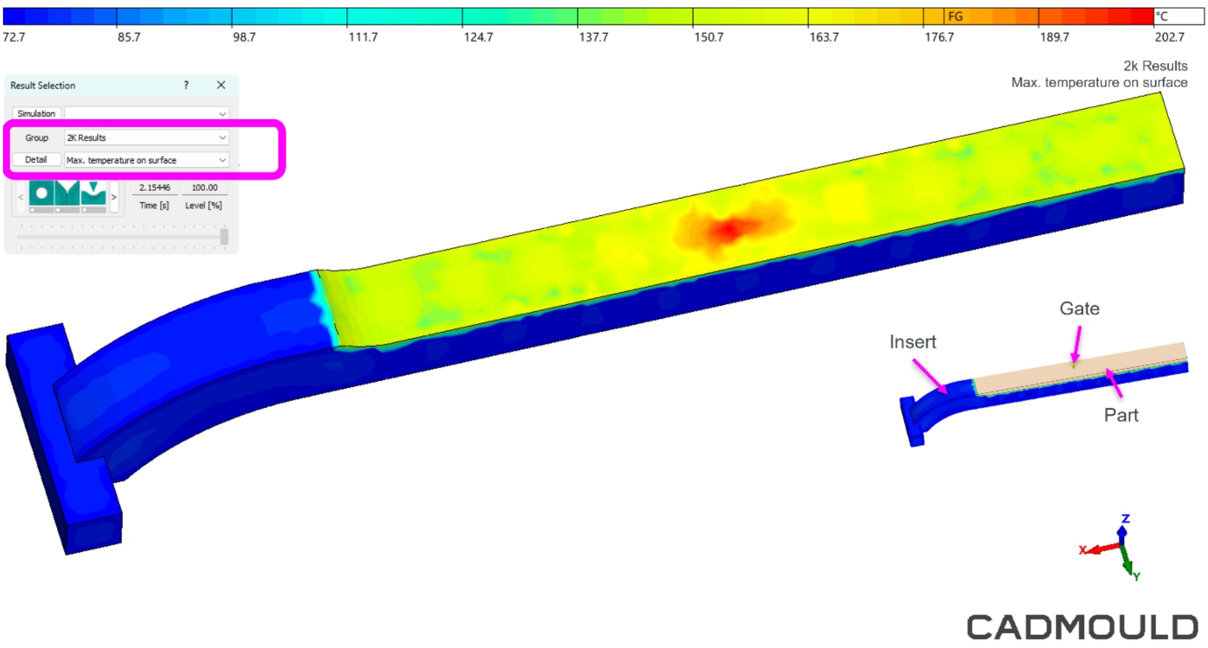

***Figure 15**: Max. Surface temperature of an insert in 2k injection molding*

With many types of inserts, **correct temperature control** at the contact surfaces is necessary to ensure a **tight bond.** In these 2K processes, it is necessary for the insert to **melt** locally (contact surface with molded part). In other 2K processes, **local melting leads** to **component defects due to temperature peaks**. The new detail therefore allows the local **max. surface temperature to be viewed quickly and easily for various process scenarios.**

The **max. Shear stress over time** describes the **local maximum shear stress** for each point on the surface of the insert. The shear stress is displayed on the surface of the insert and is used for analysis on the side of the contact surface with the mold wall and with the molded part (with the molded part hidden) (see **Figure 16**).

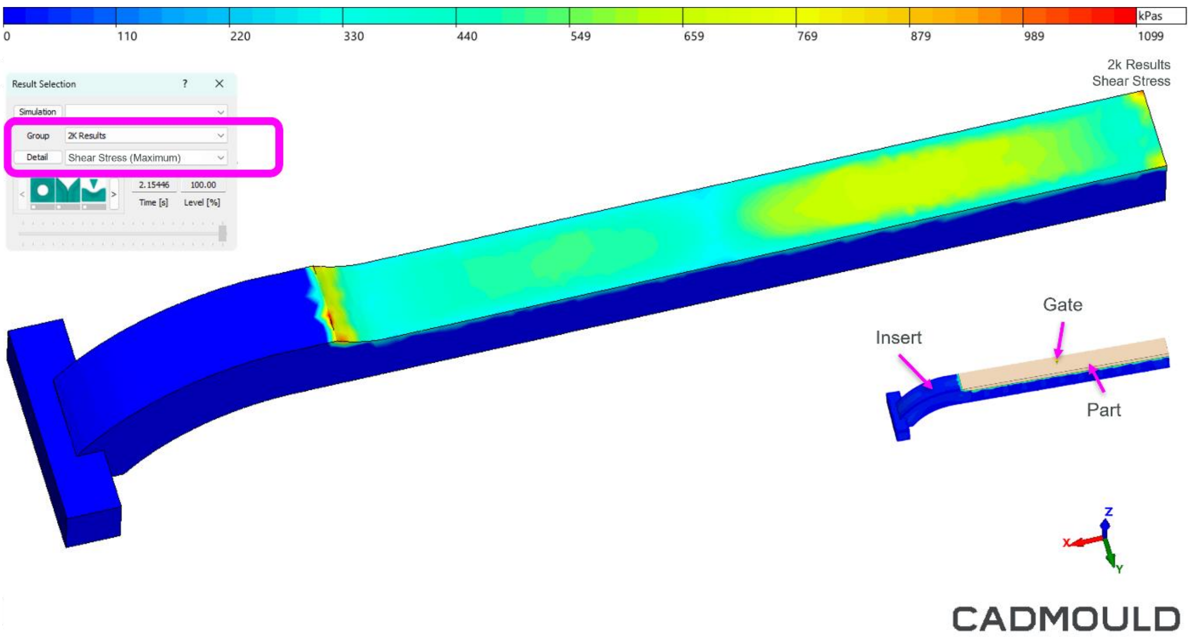

***Figure 16**: Max. Shear stress on an insert in 2k injection molding*

Insert deformation enhanced by new mechanical constraints

The injection molding process allows the direct integration of additional components, such as metal pins or other inserts. However, multi-component injection molding (2K) is also one of the increasingly used processes. A critical factor here is the possible deformation of the inserts, e.g., the first component, during overmolding or backmolding. With the help of CADMOULD, the deformation of the inserts can already be simulated. Up to now, an automatic, mechanical constraints for the inserts was used. In CADMOULD 17.1, **extended mechanical constraints** have been **introduced**, which now also allow more complex scenarios to be mapped realistically.

The mechanical constraints define the **degrees of freedom of the nodes** that are not in contact with the molded part but are in contact with the mold**.**
Following definitions are possible:

- Fixed
- No boundary condition (free movement)*
- Movement away from the mold wall*

 * New in CADMOULD 17.1

Previously, it was assumed that nodes in contact with the mold were **completely fixed.** In CADMOULD 17.1, **additional degrees of freedom** can now be defined to allow displacements away from the mold. This is shown in the following example (**Figure 17**).

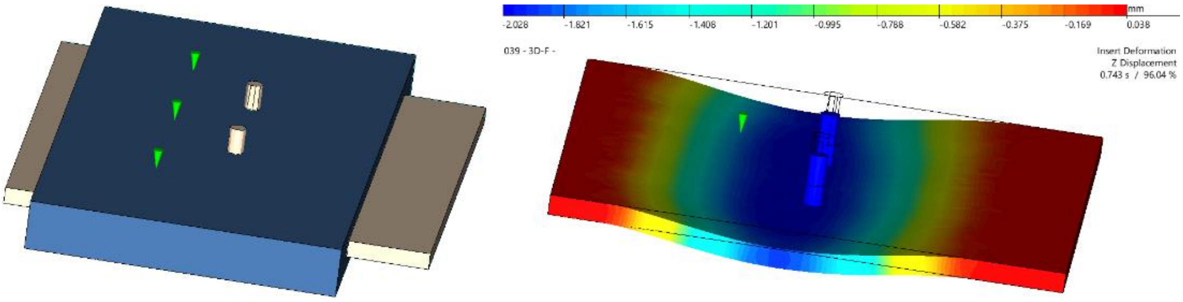

Figure 17*: Example of the deformation of the inserts in V17.1 (blue: molded part, gray: insert)*

The settings for the insert deformation are no longer in the simulation options but can now be found in the **Simulation** ribbon under the new **Insert Deformation** button.

Detailed instructions on how to use the insert deformation and **some examples** can be found at the following link:
http://www.sim-con.com/cadmould-download/V17.1/docs/Insert-Deformation_V17.1.pdf

Programming interface API: More functions controllable via script

An **API** is an interface for performing functions without opening the actual user interface. This allows you to control software commands via a **script** (e.g. Python). This opens many new possibilities, such as **interfaces** to other programs/tools and the **automation** of recurring processes. Step by step, further functions will be activated in
CADMOULD that can be controlled via API. After the first functions were made available in V16.1 and V17.0, further functions are now available:

- Converting single or multiple result files (.car) into .txt files
- Exporting fiber orientations (.cofx and .cfex)
- Generate a report in which the wall thickness of a CAD component
- Exporting deformed geometries (amplification factors
- configurable; via original network or HQ network)
- Creating HQ meshes
- Moving cooling channels
- Creating hot runners

- Import molded part**
- Mesh molded part with standard values**
- Set injection point using coordinates**
- Select material**
- Set standard process parameters for filling**
- Set standard process parameters for filling, packing and cooling **
- Load process parameters from another, existing project file**
- Save simulation with title and ID**

  ** New in CADMOULD 17.1

You can use these functions to write your own scripts, for example to:
- Optimize the **cooling channel position** automatically in conjunction with VARIMOS,
- **Automatically generate** complex **hot runner systems** according to a defined pattern and vary the connection and diameter depending on the input parameters,
- Perform **automated component thickness analyses** outside the CADMOULD environment.

If you are interested in using these innovative functions, please feel free to **contact** us. We will provide you with the functions and will be happy to support you in implementing them. We also welcome your ideas and requests for additional functions that you would like to control or automate using scripts. Your feedback can be considered for further developments. Please contact our support team **(support@simcon.com)**.

Reporting

The report generator already introduced in CADMOULD 16 has been further developed and expanded.

The first adaptation is an extension of the existing PowerPoint report templates to include a new slide: this is located directly behind the title slide as the second page in the presentation. The aim of this slide is to provide space for a summary of the most important findings or results of the simulation. This slide is initially hidden in PowerPoint and can be shown and then used if required.

Furthermore, in the PowerPoint template area, it is now possible to create **user-defined tables** with a few selected results and the corresponding numerical values and insert these into your PowerPoint templates. These tables are then automatically filled in when a report is created and serve, for example, as an overview of the most important results of the simulation. Below you will find an overview of the placeholders that can be used for such tables in PowerPoint:

Minimum flow front temperature	[#FlowFrontMinTempWhenFilled#]

Maximum flow front temperature	[#FlowFrontMaxTempWhenFilled#]
Maximum filling pressure	[#FlowFrontMaxPressureLoss#]
Maximum shear stress of the yield front	[#FlowFrontMaxShearStress#]
Minimum velocity of the flow front	[#FlowFrontMinVelocityWhenFilled#]
Maximum velocity of the flow front	[#FlowFrontMaxVelocityWhenFilled#]
Maximum temperature during filling	[#MaximumValuesMaxTempFilling#]
Maximum shear rate during filling	[#MaximumValuesMaxRepShearRate#]
Minimum sealing time	[#MinSealTime#]
Maximum sealing time	[#MaxSealTime#]
Maximum freezing time	[#MinFreezeTime#]
Maximum freezing time	[#MaxFreezeTime#]
Maximum demolding time	[#TimeOfEjection#]

FOAM: Adjustments to the nuclei per unit count for fiber-reinforced materials

In foam injection molding, an increased initial bacterial count can be assumed for fiber-reinforced materials, as these arise at the fiber ends. For this reason, the nuclei count for corresponding materials was increased from 10.000 to 25.000.

Cooling from CAD Conformal

The **Cooling from CAD** function, which has already been improved in CADMOULD V17, makes it easy to create cooling segments based on CAD files. This function assumes that bores are available and creates the segments from this.

In the case of conformal cooling, in particular with cooling channels that are additively manufactured or milled into a plate, the tempering channels are often not straight, but shaped according to the contour of the component. This type of cooling system can now also be converted into segments with a single mouse click.

To do this, create the **cooling object** as usual and assign the **HCS Component** property to it. Now you can start the conversion to segments by **right-clicking** on the object in the geometry explorer (see **Figure 18**).

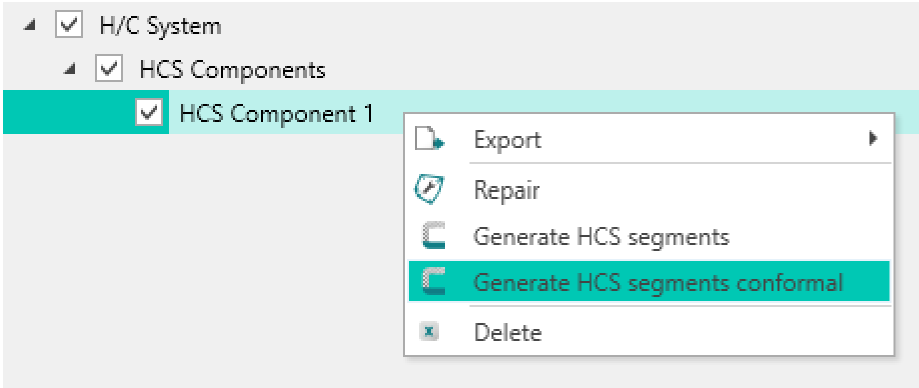

***Figure 18**: Context menu: Generate HCS segments conformal*

A new feature at this point is the option to choose between **Generate HCS Segments**, which is optimized for bores, and **Generate HCS segments conformal,** which is optimized for conformal cooling.

If you have mixed cooling systems, we recommend dividing them into different objects and converting the sections individually.

Further improvements and bug fixes

- Update to HOOPS Exchange 2024
- Troubleshooting: Manual wall temperature table selection not possible. This error has been fixed
- Bug fix: Software crash when defining unrealistic entries in runner dialog. With unrealistic (mathematically impossible) entries of the runner radius, it could happen that CADMOULD crashed. This has been fixed and the radius gets automatically reduced to a possible size.
- Bug fix: In certain situations, it could happen that the WT mode is not accepted correctly. Example: Despite the selected **constant WT**, CADMOULD assumed that cooling should be considered.
- Bug fix: When adding a second component from a different directory, the project directory was changed without being asked.
- Bug fix: No query of the project name when adding: When adding another component, no project name was queried. It could happen that unwanted old results were overwritten.
- Bug fix: Crash if paths are too long: When calculating multiple, consecutive VARIMOS variants, it could happen that the path lengths exceed the system limits and CADMOULD crashes. This error has been fixed and CADMOULD now indicates that the system limits have been exceeded.

New Features V17.1.1

Automatic variants could lead to inconsistencies

If variants in a VARIMOS project were not created in the variation window but automatically by the solver, this could lead to inconsistencies during later editing. These inconsistencies manifested in that the model creation could not be started from the parent project. Additionally, values could be displaced upon reopening. This issue has now been fixed, and projects created in the background are handled correctly.

Duplicate "settings" tab after switch

Switching the language settings during the results display previously caused two "Settings" tabs to be shown. This error has been fixed, and the unwanted duplication has been removed.

Cascade behavior from old projects

When old projects with cascade control were imported, there could be a different cascade behavior. The new parameter "Offset" was not always set to 0 in these projects, causing delays. Now, the offset time is always set to 0 when importing older projects, maintaining the original behavior.

Potential crash when creating beamer-segments fixed

Creating new beamer-segments could sometimes result in a crash. This issue has been resolved, and segments can now be created as usual.

Custom variables with relative expression

Creating custom variables in VARIMOS with relative expressions could sometimes result in an error message indicating that the range cannot be 0, despite reasonable inputs. This evaluation error has been fixed, and the error message will now only appear if there is truly an input error.

Valve gate nozzle control and prepared calculation

With certain combinations of valve gate nozzle control and prepared calculations, the calculation could fail in the background while the interface displayed an active calculation. In this case, the calculation could not be stopped, and the software had to be restarted. This behavior has now been fixed.

HTML report crashed

Using the HTML report could sometimes result in crashes, preventing the report from being generated. This issue has been resolved, and the HTML report generation is now fully functional.

Calculation of 2K with residual Stresses

Activating the output of residual stresses in the 2K calculation could result in the error message "Shrinkage_and_Warpage()=1!". In this case, no residual stress results were written. This issue has been fixed, and the results are now available as expected after the calculation.

Resetting weightings in VARIMOS

Optimizing or changing the sliders automatically reset the weighting to the default value for all quality features. This issue has been fixed, and the weighting now remains as set by the user.

Description text for sink marks in HTML report

The description text for sink marks in the HTML report has been updated according to current calculation methods. The original wording could lead to misunderstandings.

Prepared calculations via server client

The algorithm for offering and using prepared calculations has been improved. Previously, missing files that were not needed were sometimes noted.

Improved runner mesh

Creating short runner segments with different diameters at the ends could sometimes result in wrong diameters in the mesh. The meshing has now been corrected, and the results match the corresponding geometry.

Display of contact area

In calculations with inserts, contact between components is determined. This can be controlled in the simulation options. By changing the coupling distance, corrections of the coupling can be made in the case of inaccurate construction. However, reducing the coupling was not visible in the dialog. Now, the display is updated here as well, so the currently set coupling is always displayed.

Bitrate remains

Previously, changing the bitrate in the animation options was only temporarily saved and was discarded after saving the project. Now, the setting remains permanently.

Font is not remembered

Display fonts can be set individually. However, recalculating the display, e.g., by changing the window size, reset the settings. This issue has been fixed, and the set font now remains permanently.