veoCAST User Guide

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Contents

Chapter 1

Introduction

Teraport veoCAST is a tool for analyzing geometric data in terms of a casting accuracy. The application provides a process chain of semi-automated calculation steps. *Teraport veoCAST* is based on *veoBASE*. There is also a manual for this application, which you will find in the help content. Many hands-on videos already exist for numerous of analysis functions. Visit the *Teraport-Channel on YouTube*.

1.1 The veoCAST Workbench

The *veoCAST Workbench* is the central calculation interface for casting analysis. It is structured in a way that all subject areas are processed via their own tabs (for example maxima or minima analysis). Each tab contains the required user interface for one analysis step. Because of analysis steps can be depended from each other, it is recommended to work trough the tabs from left to right.

Each topic is treated in detail in the *veoCAST* help, subchapter for the respective workbench. At the top right corner of each workbench, the icon *"Help on the selected tab"* appears. By clicking on this icon the *veoCAST help* will open automatically for the respective subchapter. The subchapter Assembly preparation describes how to start the workbench.

1.2 Export of result geometries

If the new result geometries are created by an analysis method, then these can be exported (with existing format license) to the following exchange formats: JT, STL, WRL, TP. Individually selected result geometries can be written to any directory via the context menu entry *"Export result"*. Existing files with the same name will be overwritten during export. The function *"Export all results"*, in the upper right corner of the Workbench, writes all results of the currently active tab into a single structured file.

Chapter 2

Assembly preparation

The following text explains all steps required to start a geometry analysis.

- **Load CAD file**

  First, a CAD file must be loaded. In the *Assembly tree* on the left side, select the folder *"Assembly"*. Open the context menu with the right mouse click and select *"Add Part(s)"*. In the file browser, select the desired CAD file. Activate the check box next to the name of the CAD file in the *Assembly tree* in order to display the geometry. To focus the view on the loaded CAD geometry, click the *3D View* and press *"Ctrl+A"*. (see manual *DMU.View*, chapter *"Loading of components, structures and paths"*)

- **Selection of the components**

  If the CAD file contains only the component to be analyzed, proceed to the next point.

  If the CAD file contains unwanted components, these must first be hidden. To do this, open the part structure in the toolbar via *"Open Part Structure"* . On the left side, the tab *"Part Structure"* opens, showing the CAD geometry as a tree. In the *3D view* you select with the left mouse click the part to be analyzed. This will be colored red. Use the context menu (right mouse click) to select *Hide others*. If the component you want consists of several components, hold down *"Ctrl"* while selecting the components.

  Alternatively, you can hide or show components via the tree view in the *Part Structure* tab.

- **Start of the geometry analysis with veoCAST**

  Select the CAD geometry in the *Assembly tree* inside the folder *"Assembly"*. Right-click to open the context menu. Here you select *"Veo"* and select in the submenu veoCAST. This will open the *veoCAST Workbench*. Alternatively, select the component and select the veoCAST via the corresponding button in the toolbar. Fill in the data fields *"Administration"* and then carry out a detailed analysis. The subchapter *Analysis tab* describes the tools available.

Chapter 3

Analysis Tabs

The workbench *"veoCAST"* bundles analysis methods for individual casted parts. It is not distinguished which casting process (injection molding, casting, gravity casting, etc.) and which material (plastic, aluminum etc.) is used. Therefore, it is the users responsibility to configure the respective limits of the analyzes (e.g., allowable wall thicknesses) accordingly. The goal of the workbench is to reduce the number of iterations required for the development of cast components and to shorten the process chain sustainably. The following subchapters describe the input fields and analysis tools (divided into tabs) in detail.

3.1 Administration

The tab *"Administration"* is used to capture administrative data. The entered data is saved as text format. Analysis date is automatically suggested as the current date. You can manually override it or use the calendar widget on the right. Contact Us Teraport if you need changes or additional data fields for your administrative data.

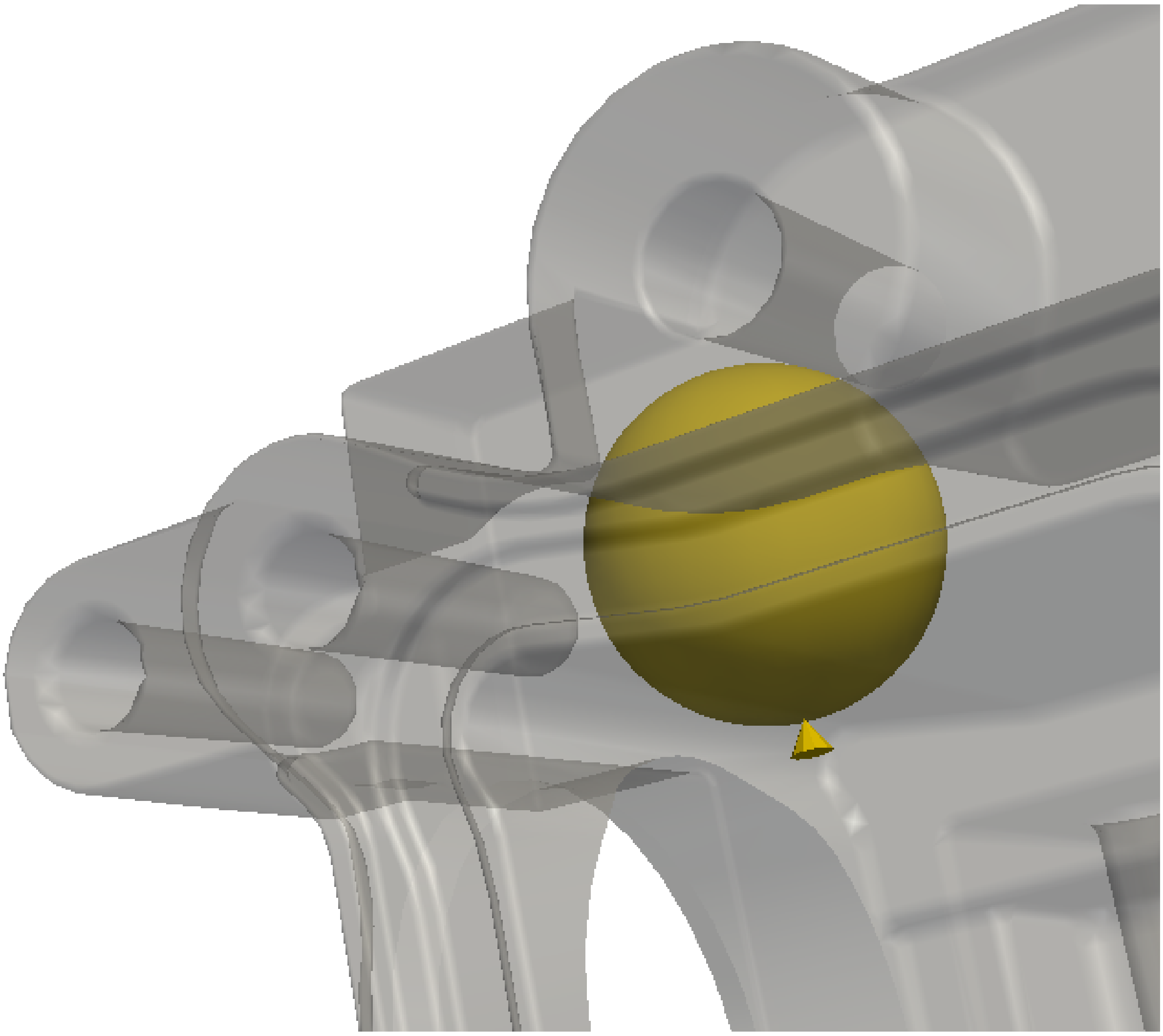


Figure 3.1: Representation of *Thickness-maximum* as a sphere

3.2 Maxima

The goal of the maxima analysis is to uncover possible accumulations of material. The maximum wall thickness in *veoCAST* is defined as the diameter of the maximum inscribable sphere (based on the so-called "circle method" according to Heuvers). The locally thickest surface point is represented as a sphere which lies completely within the geometry. In addition, a cone is displayed on the parts surface. The cone points towards the center of the sphere.

The parameter *Raster size* indicates the size of the search cells for which a maxima analysis is performed. At most, one thickness sphere is generated per search cell. Please note that if the *Raster size* is too large, not all value overruns may be displayed. A small *Raster size* causes a more accurate analysis while increasing the computing time.

The *Minimum thickness* defines from which diameter thickness spheres are displayed. Smaller diameters do not appear in the 3D-View or in the results table. If you have small wall thicknesses, you may also need a small *Raster size* to avoid missing the value. The *"Start analysis"* button executes the maxima calculation. Depending on the part size, complexity, and *Raster size*, the analysis may take several minutes. A re-analysis can only be configured and started if all existing results have been deleted with *Reset analysis*.

3.2.1 Color coding of the maxima spheres

The generated maxima spheres are automatically assigned with a color value. Balls with the thickness value close to the *Minimum thickness* defined by the user get the color "blue", the sphere with the maximum thickness found in the part gets the value "red". For intervening thickness values, the color value is interpolated:

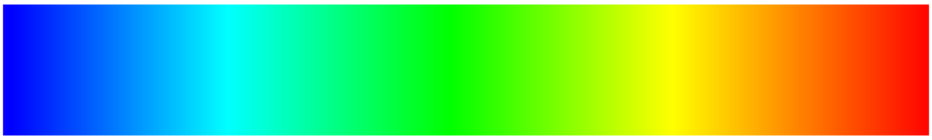


Figure 3.2: Color coding of result values (from minimal to maximum)

3.2.2 Functions of the result table

3.2.2.1 Sort

After completion of a maxima analysis, the thickness spheres found are listed in the results table below the control panel. By clicking on the respective column header (e.g., "Thickness") you can sort the results by the selected value. Click again to reverse the sort order.

3.2.2.2 Create cutting planes

In the context menu (right mouse click) for each thickness sphere you will find the entry *"Create section plane by element"*. This function places a new cutting plane through the center of the selected thickness sphere. For all existing planes the function "*3D cutting*" is deactivated, all other thickness spheres are hidden. The layer is attached to the selected maximum in the table, and you can use the visibility to activate or deactivate the plane. To move, rotate or mirror the plane, use the menu *"Open Properties"* (see manual *veoBASE*, chapter *"Open Properties"*)

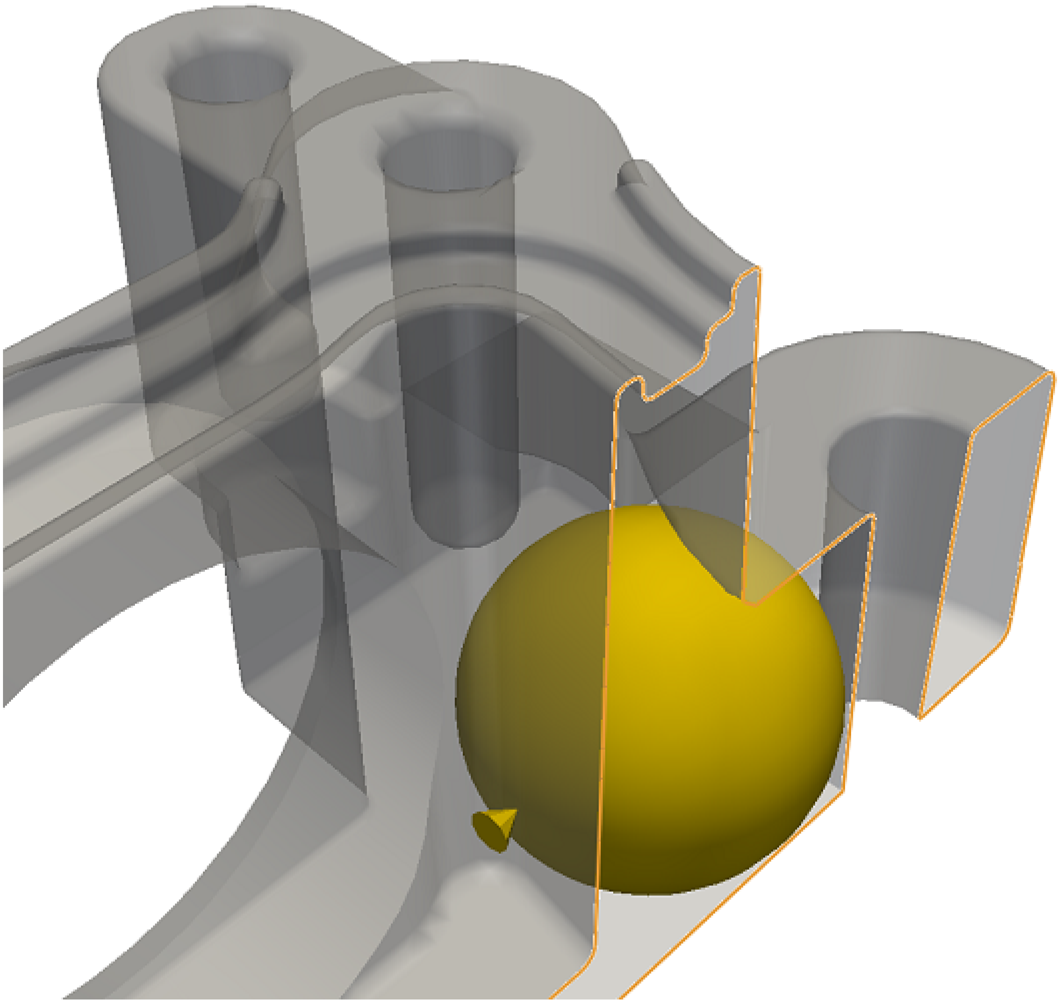


Figure 3.3: Section plane through a *thickness-maximum*

The context menu entry *View result by section plane* zooms the generated section plane so that it considers the maximum orthogonal. Shortcuts are available for both functions, allowing fast editing.

3.2.2.3 Align cutting plane

You can use the context menu entry *Align cutting plane* to specify the alignment of the created cutting plane.

- The entries X, Y and Z align the plane orthogonal to the main axes.

- *Measurement* the plane passes through both points of the maximas (midpoint, surface point).
- If *Triangle* is selected, you must select a triangle of the part surface that defines the plane in the 3D-View.
- *Reverse direction* flips the cut direction, e.g. +X and -X.

3.2.2.4 Look at result

In the context menu select *View Result*: the 3D-View zooms to the selected thickness sphere in a way that the center of the sphere is the new pivot point of the 3D-View. With the shortcut *"CTRL+F"* you can call the function for a selected row in the result table.

3.2.2.5 Visibility

Use the *Visible* column to show or hide individual results. Alternatively, you can select *Show Selected* or *Hide Selected* in the context menu (even for several lines) to control the visibility. When switching to another tab, e.g. *"Administration"*, all thickness results are hidden, when returning to the tab *"Maxima"* they appear again.

3.2.2.6 Rating

For each result, you can define how critical this value for manufacturability is. First, the rating of all lines is set to *"Unweighted"*. You can manually assign *"Critical"* or *"Uncritical"*. This will allow you to check if you have checked each row of the result table.

Possible problems with thickness calculation

- A cone is not visible when inside a hole or covered by geometry.
- Parts with very large dimensions and at the same time very small wall thickness can not be processed.
- Only one thickest spot is displayed per grid cell. For parts with similar thicknesses, the position of the ball is not unique.
- Hollow bodies with very small openings to the outside may be mistakenly considered as a volume object.
- Parts with large mesh errors (holes) may not be recognized as a solid object.
- Inner geometry (also from modeling artifacts) is not ignored. As a result, the *Maximum thickness * may not be determined correctly, so testing for internal geometry is necessary.
- For very complex parts, only a *Local maximum thickness* may be found if the raster size is too large.

Thickness determination by manual measurement

In addition to the automatic thickness measurement *vepBASE* offers the ability to manually determine the thickness. To do this, select *"Activate Thickness Measurement"*. Now thickness measurements can be performed directly on the parts surface. Thickness can only be measured between two nearly parallel surfaces.

3.3 Minima

Depending on the manufacturing process, parts must have a certain minimum wall thickness. Otherwise, the part is not suitable for pouring and solidifying. Therefore, thickness drops can be determined in the tab *"Minima"*. A local minimum is defined here as the smallest two-point measurement between two nearly parallel surfaces in a grid cell. The angle tolerance between two surfaces uses a constant value (45 degrees). The midpoint of the two-point measurement must be clearly within the material (solid test).

Similar to calculating the maxima, the *Raster size* can be used to determine how many results will be displayed. For each search cell, at most one thickness sphere is displayed. If adjacent results are closer than *Raster size*, only the smaller value will be retained. Furthermore, the triangles of the part surface are divided during the analysis in such a way that the maximum edge length corresponds to the *Raster size*. As a result, you should use a small *Raster size* to ensure that you can see any thickness underruns.

The *Edge filter* eliminates the effects of minima that occur close to the part edges, such as tapered ribs. The higher the value is, the more results are filtered out. If the value is set to "0", the distance to the edge is ignored. With larger values for the *Edge filter*, it is possible to find places where separate "Chambers" (for example different pressure areas, tubes) lie in an inadmissible close proximity.

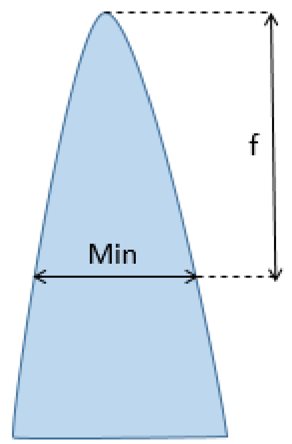


Figure 3.4: Checking the minimum distance "f" to the part edge in the *Edge filter*

With the *Lower bound* you exclude very close parallel surfaces from the result. The *Upper bound* determines to what value wall thicknesses are marked as "too thin".

Possible problems with the minima calculation

- For tapered, nearly parallel geometry, a location near the *Lower bound* is found.
- For angled or rounded geometry, the surfaces that are "just still parallel" are found.
- Calculation requires solids, so non-repairable tessellations (holes) can cause problems.

Thickness determination by manual measurement

In addition to the automatic thickness measurement *veoBASE* also offers the ability to manually determine the thickness. To do that, select *"Activate Thickness Measurement"* in the toolbar. In the 3D-View you can now perform thickness measurements.

3.4 Gaps

The mold with which a cast is produced is a geometric "negative" of the part. Consequently, close surfaces on the part produce very thin walls at the mold. These thin walls do not have sufficient strength to consistently produce high quality parts. The analysis tab *"Gap Dimensions"* checks whether sufficient distance between walls are maintained. Detected are close, near parallel walls. The angle tolerance between two surfaces is a constant value (45 degrees). The smallest two-point measurement between two nearly parallel surfaces in a grid cell is displayed.

Similar to calculating the maxima, the *Raster size* can be used to determine how many results are displayed. For each search cell, at most one gap drop is shown. If adjacent results are closer than *Raster size*, only the smaller value will be retained.

The *Edge filter* eliminates gap measurements close to shallow depressions, such as seals for gaskets. The higher the value of the *Edge filter* is, the more results are filtered out and the deeper the columns have to be, see picture. If the value "0" is set, the distance to the bottom of the gap is disregarded.

With the *Lower bound* you exclude very close parallel surface from the result. The *Upper bound* determines up to which value gaps are marked as "too thin". The gap below (minima) are represented by two inwardly directed cones, which are connected by a line. The user-defined *Upper* and *Lower bound* defines minimum and maximum color as shown in subchapter Color Coding, i. the smallest gap is colored "red", the largest "blue". For the gap measure results, all functions of the result table already described in chapter Maxima are shown as e.g. Cutting plane and focus function etc. available.

- Tip: To see the location of a gap undercut on the part, use *"Show Pivot"* from the view options of *3D View*.

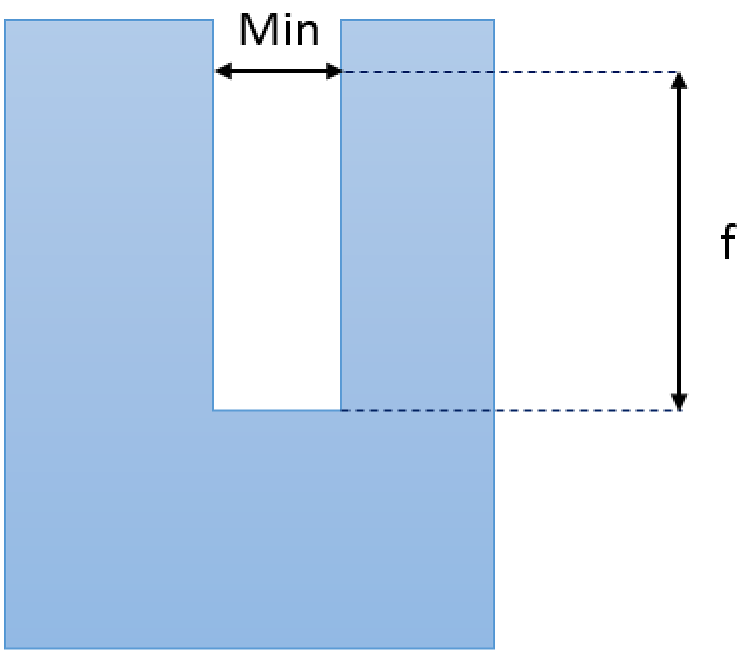


Figure 3.5: Checking the minimum distance “f” to the underside of the gap in the *Edge filter*

3.5 Isosurfaces

The analysis in the tab *Isosurfaces* visualizes areas with the same material thickness: Each point on an isosurface has the same distance to the part surface. Several isosurfaces can be created and displayed at the same time. The advantage of isosurface analysis compared to a *Maxima* analysis is that not only the center of a material accumulation is marked, but the entire material distribution in the part becomes visible. It shows whether there are accumulations of material, constrictions or jumps of the wall thickness. The material flow and the cooling behavior can be estimated without a CFD simulation.

The analysis is divided into two steps: preprocessing and isosurfacing.

Preprocessing

Initially, the geometry data is pre-processed with a user-specified resolution. The lower the value, the more accurately the geometry is analyzed. At the same time, for small resolution values the calculation time and memory requirements increase.

- If the value used for *Resolution* is too small, the main memory may not be sufficient and the error message *“Calculation failed”* will appear.
- If the value used for *Resolution* is too large, the isosurface position is heavily error-prone and differs significantly from the results in the tab *“Maxima”*. Depending on the found depth values, in extreme cases even no isosurface can be created. The message *“Calculated data structure not usable”* will be displayed. For a meaningful evaluation, the *Resolution* should be significantly smaller than the maximum wall thickness, e.g. factor 10.

Isosurface

After completion of preprocessing, the slider is activated. Number fields located to the left and right of the slider indicates the limits for which isosurfaces can be generated. In the centered numeric field, the currently selected value is displayed. This value can be changed by moving the slider or by manual input. Choose a distance value for which an isosurface should be created, e.g. 10mm, and press the button *"Create surface"*. In the result table and in the 3D-View, a new isosurface appears with the selected value. The numeric values displayed to the left and right of the slider define minimum and maximum colors as shown in subchapter Color Coding, i. the surface with the smallest value is displayed "blue", the surface with the largest value "red". To automatically generate multiple isosurfaces with increasing value, use Quick Check.

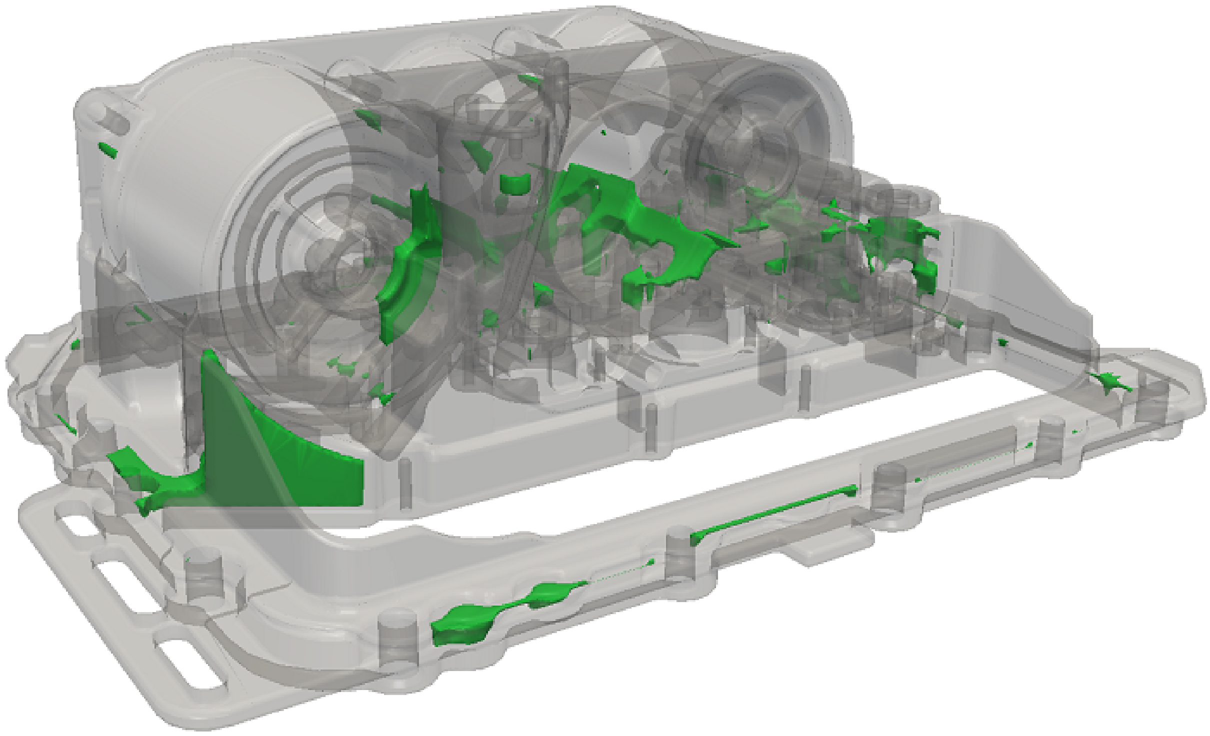


Figure 3.6: Transparent part with isosurface

The surface with a 10mm iso-value has a 5mm distance from the parts surface sides - for a 10mm thick wall it should be exactly in the middle (Due to the limited *Resolution*, in this case, you should create an isosurface with 9.5mm for visualization.) For newly created isosurfaces, colors and transparency values are automatically predefined. You can, however, subsequently define colors and transparency values per surface, see (see manual *veoBASE*, chapter *"Open properties"*)

If several isosurfaces have been created, you can use the slider to quickly switch between areas: Move the slider to see the area closest to the current iso value. All other surfaces are hidden. To better analyze the interior of a part, you can use different tools to create sections, as shown in the figure (see manual *DMU.View*, chapter *"view cuts"*) .

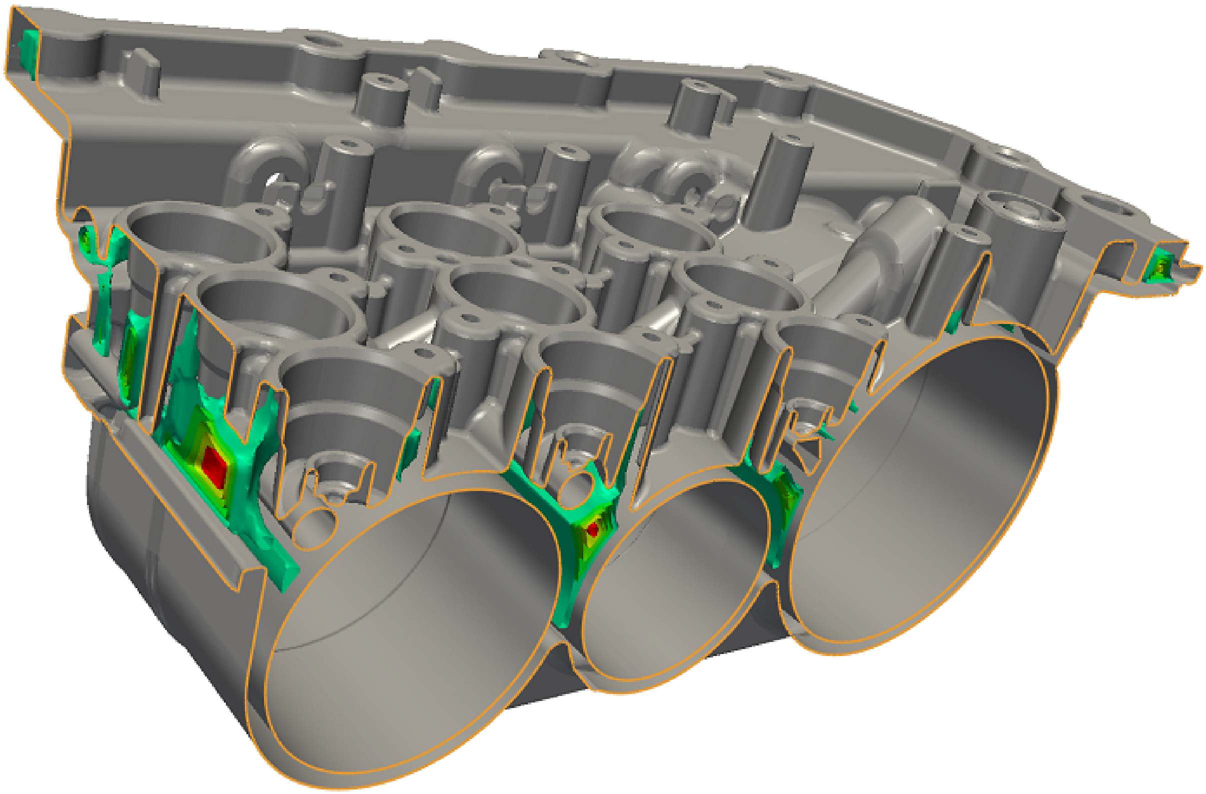


Figure 3.7: Part with several isosurfaces and two cutting planes

Possible problems with the production of isosurfaces

- Hollow bodies with very small openings to the outside may be mistakenly regarded as a volume object.
- Parts with large triangulation errors (holes) may not be recognized as a solid object. In this case, isosurface analysis is not possible.
- The generated isosurfaces are an approximation of the ideal isosurface. Due to the discretization and value interpolation the isosurface deviates from the ideal position. Decrease the value of *Resolution* to produce a more accurate representation.
- Avoid creating isosurfaces near the part surface. Due to implementation details, larger display errors can occur in this case, so that false surfaces are even visible outside of the part. Calculate with a finer resolution and generate surfaces further inside the part to exclude these errors.

3.6 Depth Scopes

The tab *"Depth Scopes"* shows the depth of the part color-coded on the parts surface. The depth value indicates the diameter of the maximum inscribable sphere that touches the colored surface point. First define the calculation parameters and then start the calculation. Hide the part after the calculation in the assembly tree to better recognize the depth ranges. The found depth values are subdivided evenly into 10 differently colored color areas. These can be edited afterwards (color, value range, number). You only have to *"Reset Calculation"* if you want to change the calculation parameters.

The parameter *"Accuracy"* determines how precisely depth is measured and how often the surface is subdivided. For fast and rough calculations, choose the values between 0.5mm and 1mm. To analyze small, filigree parts or to create a surface depth with improved accuracy, select the 0.05mm to 0.1mm.

The parameter *"Max. Depth"* is used to analyze thin wall thicknesses faster and more accurately. The depth determination stops as soon as the entered value has been exceeded, resulting in a shorter computing time. The generated value range is cropped and the resulting color fades through the smaller value range on, so that thin wall thicknesses are better distinguishable.

With the parameter *"Sharp Edges"* the depth measurement is changed in such a way that edges do not produce a low depth value. The higher the *Sharp Edge* value is, the harder(sharper) edge will be tolerated. Note that the value can cause small measurement errors. For the most accurate depth measurement, choose a small value, e.g. 0.1mm.

The check box *"Invert color gradient"* changes the color gradient from "blue(thin)-red(thick)" to "red(thin)-blue(thick)".

In the table *"Depth ranges"* you can export, remove and add individual depth ranges via the context menu (right mouse click). Color changes (via the *Color* button) are immediately reflected in the 3D view. The *"Depth"* can be used to adjust the upper limits of each color. When adjusting depth values above/adding depth ranges, it is necessary to select *"Update depth ranges"*.

3.7 Undercuts

After manual definition of a separation plane undercuts can be automatically detected. The undercuts are shown as result geometry in the 3D-View. You can then decide whether existing undercuts can be avoided, whether draft angles are necessary and whether the production and fixing of cores is easily possible. After the calculation, you can assign each undercut to a slider or an insert. The undercut analysis is divided into three sub-steps, which are described in detail in the following sub chapters.

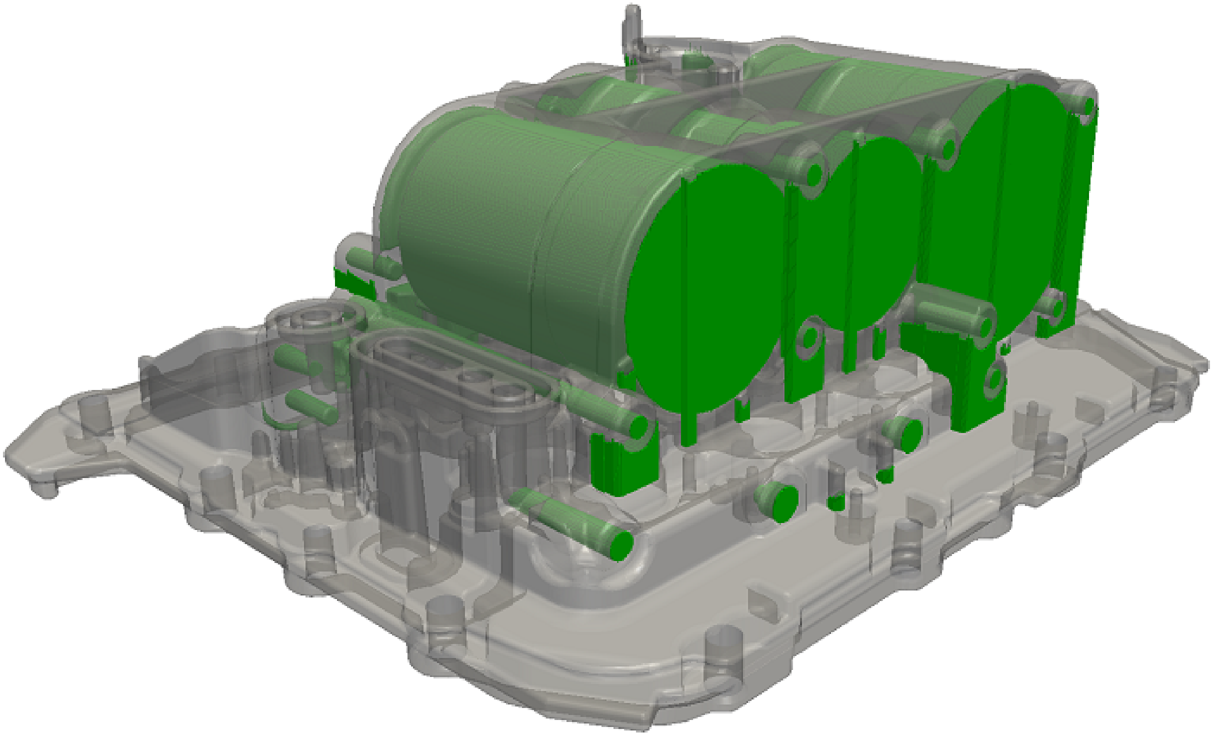


Figure 3.8: Part with visible undercuts

3.7.1 Create Separation Plane

The *Undercut Calculation* is based on the *Separation Plane*. This determines in which direction the tool halves opens. The *Separation Plane* can be created with four different tools:

- Select the tool *"3 Points"*. Three points can be set on the surface to define the separation plane. Use this tool if the part consists only of curved surfaces.
- Select the tool *"Triangle"* and move the mouse pointer into the 3D-View. The triangle below the mouse pointer is highlighted.
  Left-clicking places a plane through the selected triangle. Use this tool if the *Separation Plane* can be defined by a flat surface.
- Select the Tool *"Line"* and draw a line in the 3D-View over the part surface. A plane that is perpendicular to the part surface and the drawn line is created. This tool is useful if no part surfaces exist in the *Separation Plane* orientation, but there are side surfaces that are in *Direction*. The following example shows a part with an obliquely oriented dome and a tilted dome surface. The first step is to align the dome vertically in the view. In the *3D View*, use the key combination *"Ctrl+R"* to set the pivot point on the dome and rotate the view. In the second step they draw a horizontal line on the dome (black line in the left picture). The clamped *Separation Plane* has the perfect alignment for unmolding the dome (right figure).

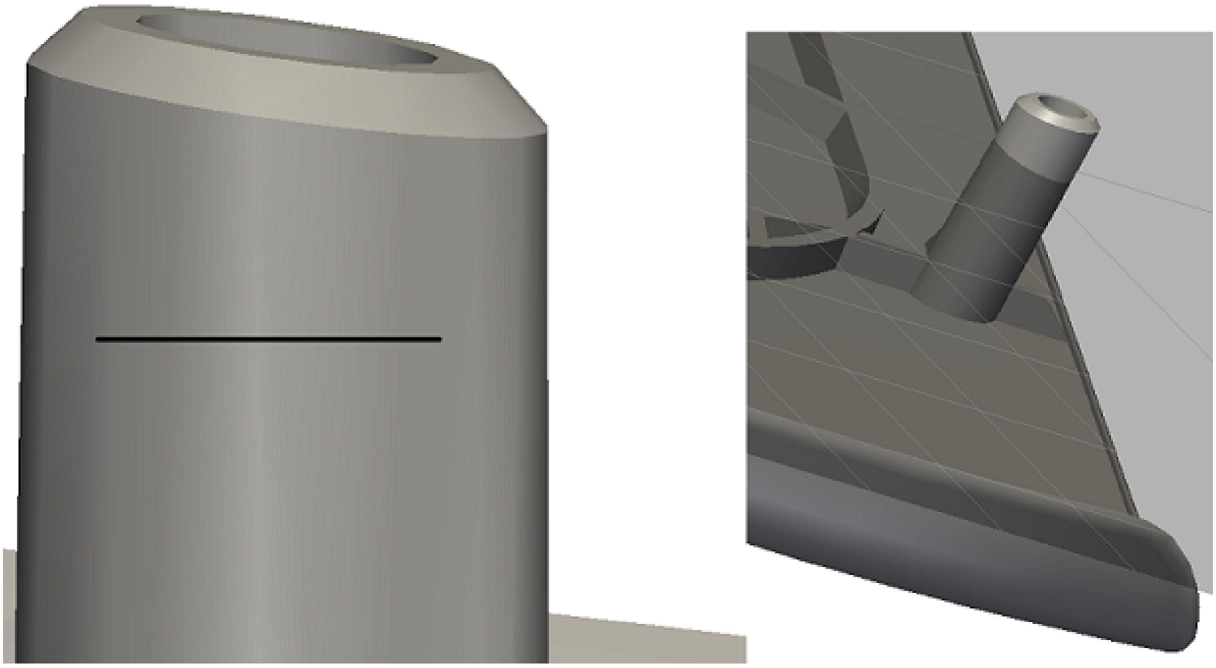


Figure 3.9: Create a *Separation Plane “Line”*

- If the CAD geometry contains the *Direction* provided by the designer, it can be read out. To do this, select the *Direction* in the *Structure view*. In CATIA, the direction of demolition is usually filed under *“Design_Work”*. Click on the button *“Direction”*. In order to be able to use a *Direction* from the geometry, this geometry must be displayed(activated). Then select the button *“Direction”*.

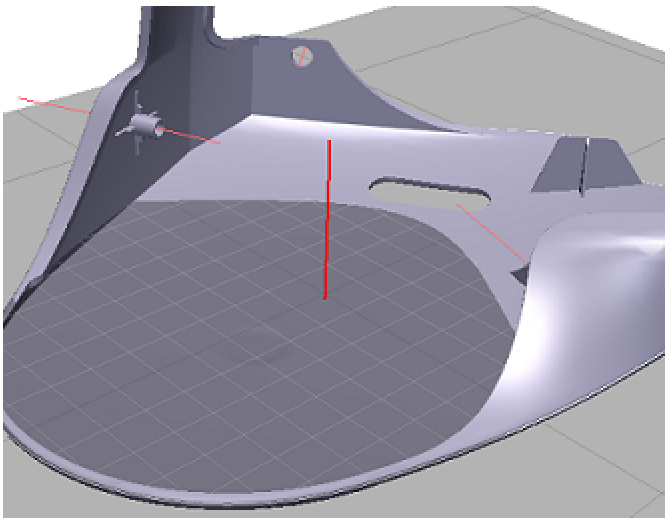


Figure 3.10: Select *Direction* in *3D View*

The plane is centered on the selected unmolding line and can be moved later, see subchapter *“Rotate and move the separation planes”*.

Note: The position of the plane is only an illustration of the tool separation. For the calculation of undercuts and bounding boxes the alignment and not the position of the *Separation Planes* is decisive. The plane does not have to be moved.

3.7.1.1 Rotate and move the *Separation Plane*

After creation, the *Separation Plane* can be rotated and moved as desired. To do this, select the *Separation Plane* in the 3D-View. Open the Properties menu and activate the manipulator as described in the manual *veoVASE*, chapter *"Open properties"*. In the *3D view* a compass appears. The plane can be rotated via the red and green arc of the compass. The blue axis is used to move the plane.

Create a new *Separation Plane*

If there is already a *Separation Plane*, you can create a new *Separation Plane* using the tools described above after *Reset Calculation*. If computations have already been performed based on the existing plane, all obsolete data (e.g., undercuts, cams, inserts, bounding box, projection, tool halves) will be discarded.

3.7.2 Undercut calculation

After creating a *Separation Plane*, undercuts can be automatically detected with the command *"Calculate undercuts"*.

Possible problems of the undercuts calculation

- Note that the *Undercuts* are determined by a discretization method, i. for very large part (>1.5m), relatively small undercuts (<1mm) may be misclassified or, in exceptional cases, not recorded.
- If there are large defects (holes in the surface) in the cross-linking of the part, the entire interior of the part is classified from undercut. In this case, a single undercut has a very large volume, but can not be seen in the 3D-View (or only at the location of the hole in the part surface). If necessary, hide the part to verify that it is. The unrecognized undercut can be removed with the command *"Delete selected part"* in the toolbar.

Undercut Categories flat and depth

All determined undercuts are first classified into the categories *"Flat"* and *"Depth"*.

- Deep undercuts are usually real problems and may need to be handled in toolmaking by *Cams* or *Inserts*. They are highlighted in green in the *3D View*.

- Flat undercuts are not displayed automatically after the calculation and can be reloaded using the command *"Show selected"* in the context menu. Flat *Undercuts* are often caused by poor CAD data quality or minimally tilted *Separation Plane*. They are marked in yellow in the *3D View*, with low *Undercuts volume* they can usually be ignored. However, flat undercuts may be an indication of missing draft angles.

Inspecting the undercuts

It is the responsibility of the user to test both *Flat* and *Deep* undercuts. All *Undercuts* can be viewed together in the *3D View* or processed step by step. To check an *Undercut* individually, select it from the *Result table * and open the context menu with the right mouse click. Select *"Reframe"*, and then the *3D View* will zoom to the selected *Undercut*. This is dyed red. By marking and inspecting several *Undercuts* you can make several or all *Undercuts* visible again.

When zooming to the *Undercuts*, it is not checked whether the *Undercut* is freely visible or obscured by the part. If you do not see the *Undercut* after *"Reframe"*, then change the perspective in the *3D View*. Alternatively, in the *3D View*, set the display mode to wireframe using the key combination *"Ctrl+W"* (**W** ireframe). Use the key combination *"Ctrl+P"* (**P** hong) to return to the normal display.

Recalculation

If you are not satisfied with the result of an *Undercut Calculation*, select *Reset Calculation* first and then create a new *Separation Plane*. All existing *Undercuts* will be discarded and can not be restored. The calculation will not be triggered automatically after changing or creating a new level, so select again *"Calculate Undercuts"*.

Sliders and Inserts

After calculation of undercuts it has to be decided how each undercut can be handled in the molding process. You can choose from *Sliders* and *Inserts*. Inserts can be created and manipulated analogously to Sliders, so in the following text only sliders will be discussed. Create a new slider by opening the context menu in the *Slider* table. If you select *"Create Slider"*, a new entry appears in the table. Proceed analogously to create *Inserts*.

Assign Undercut(s)

Each undercut can be assigned to a slider or an insert. To do this, select the undercut from the table and drag it to the *Slider*. The *Undercut* appears below the slider and is removed from the *Result table* (left). To cancel an assignment, select the undercut and select *"Delete Undercut"* in the context menu. The undercut is then inserted back into the result table. You can drag multiple undercuts into a single slider.

Adjustment of the slider's box

After assigning undercuts to a slider, a box is automatically created (shown in the 3- View as a black box), which symbolizes the dimensions of the slider. The box is aligned to the *Separation Plane* and initially represent the minimum dimensions of the undercut. Boxes can be marked and resized in the 3D-View or in the table. For resizing, the box can be clicked and adjusted on the gray spheres (see illustration). The surfaces of the box can be used for rotation. The length, width and height of the slider depends on the respective maximum value of the box contained.

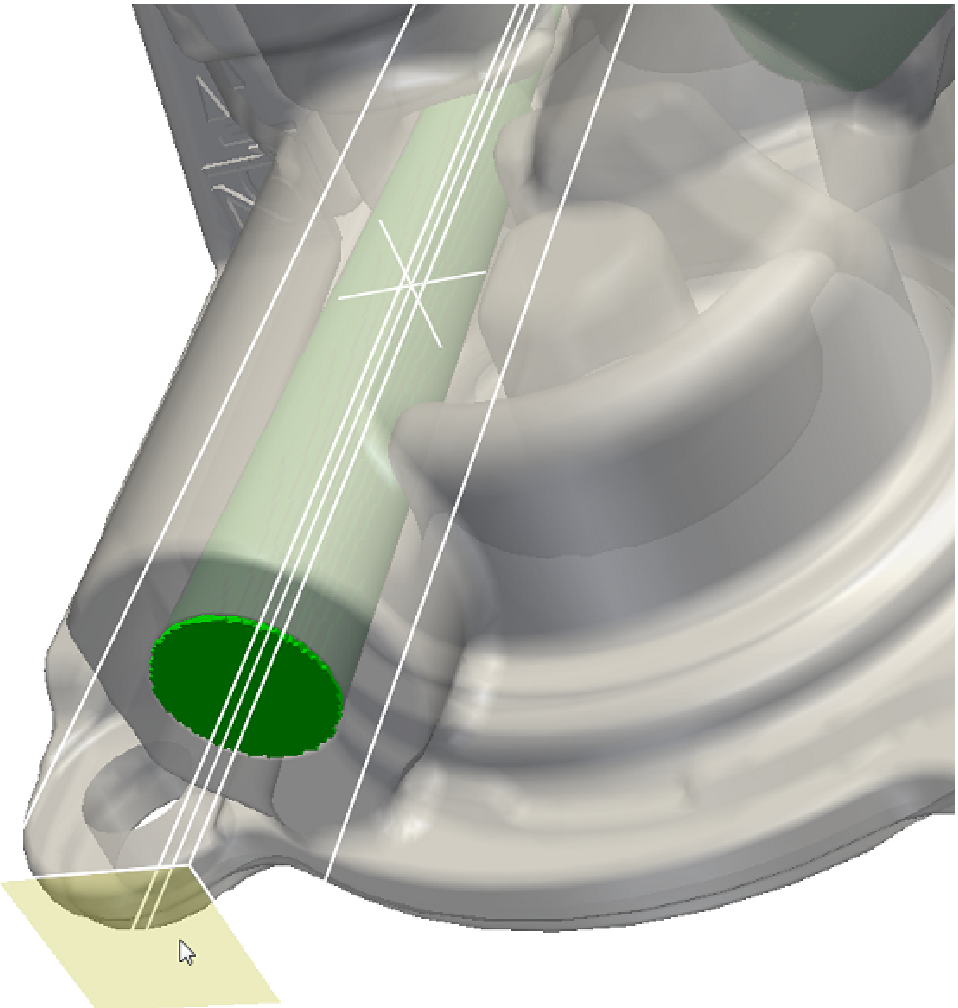


Figure 3.11: Adaptation of a *Bounding Box* for a cam

3.8 Dimensions

In the tab *"Dimensions"* you can record the spatial extent and orientation of the part using a bounding box. The bounding box encloses the entire component and is aligned by the previously defined separation plane (in the tab undercuts). To determine the dimensions, select the *"Create Bounding Box"* button . After the calculation, the smallest possible plane-oriented the Bounding Box of the component is displayed, see figure. Length, width and height are automatically entered in the data fields.

3.8.1 Product size

In the tab *"Dimensions"* you can record the spatial extent and orientation of the part using a *Bounding Box*. The *Bounding Box* encloses the entire component and is aligned by the previously defined *Separation Plane*. To determine the *Dimensions*, select the *"Create Bounding Box"*. After the calculation, the smallest possible plane-oriented the *Bounding Box* of the component is displayed, see figure. Length, width and height are automatically entered in the data fields.

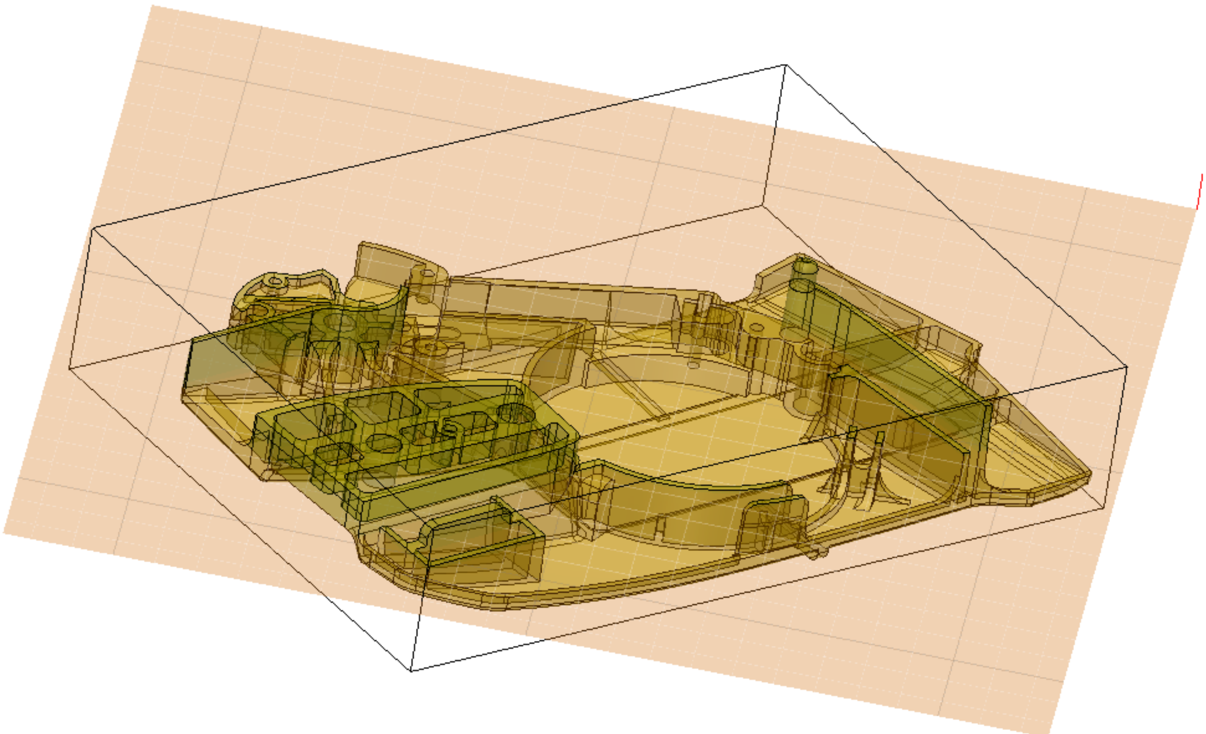


Figure 3.12: *Bounding Box* und *Separation Plane*

It is possible to rotate the "*Bounding Box*" in the defined *Separation Plane* to influence the orientation of the part in the tool. To do this, select *Manipulate Bounding Box* and move the compass on the blue arc as shown. The dimensions are recalculated accordingly. Press *"Delete Bounding Box"* and then *"Create Bounding Box"* to restore the smallest possible size.

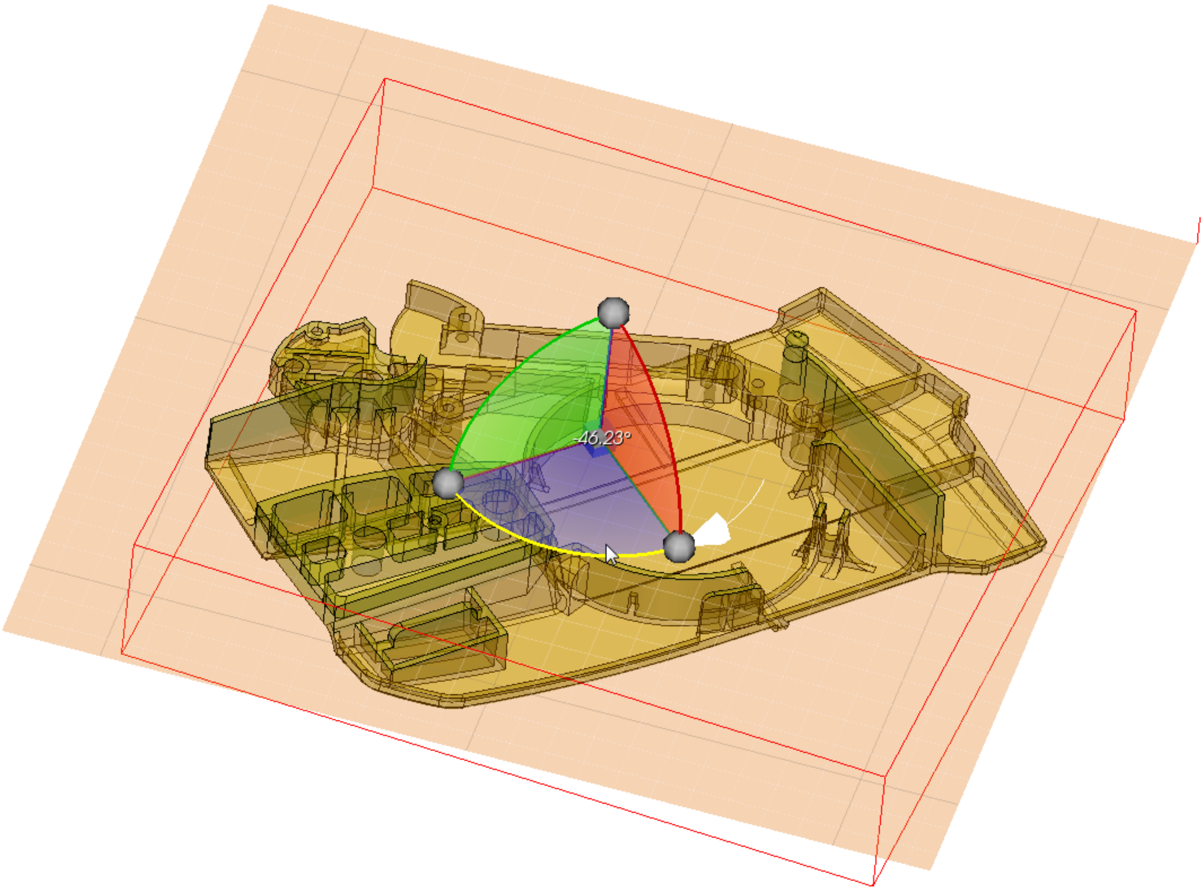
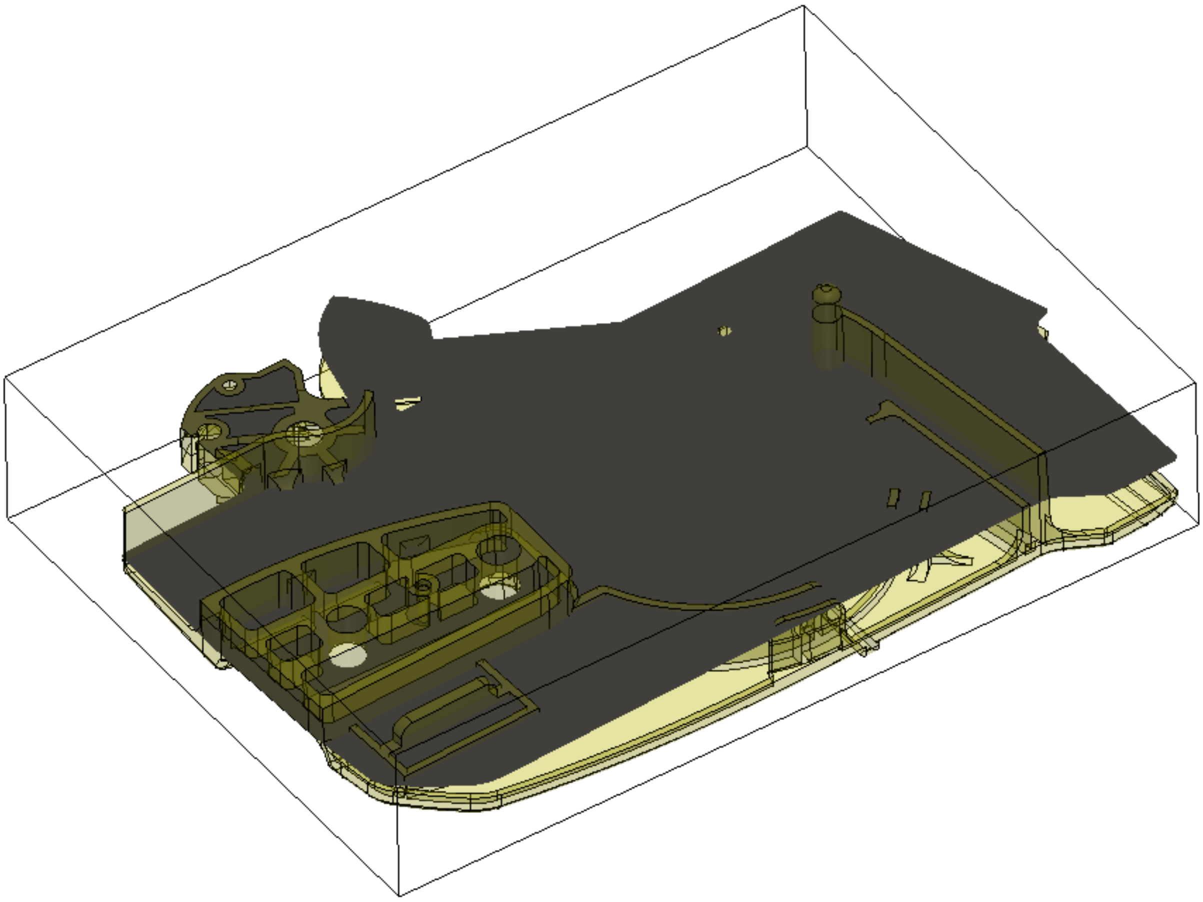


Figure 3.13: Bounding Box adaptation by rotation with the compass(selected circular arc are marked yellow)

3.8.2 Calculate of projected area

The *projected area* of the component with regard to the *Separation Plane* can be determined with the command *"Calculate projected area"* in the *"Product"* tab. After the calculation, the *Projection* in the *3D View* is visualized as a gray area below the component (see figure) and the determined area value is entered in the *Workbench*. Based on the user specified cavity pressure, the tool lift force is calculated and rounded to the nearest hundredth place. The value thus determined is used to select the appropriate tool based on the clamping force.



3.8.3 Tool Halves

To create the molding tool, an activated separation plane and an bounding box (from the tab "*Product*") are required. After selecting *"Calculate tool halves"* the tool halves will be displayed as geometry in the 3D view above. *"Delete tool halves"* removes the generated geometry from the 3D view.

3.9 Occlusion

Based on the separation plane defined in the tab *"Undercuts"*, further characteristic values of the part can be determined. After defining a separation plan, the analysis *Occlusion* is available. Set the desired draft angle (in degrees) and start the analysis.

The part surface is then classified into 6 different categories:

- Touchable from above (resting on the upper half of the mold).
- Touchable from below (resting on the lower half of the mold).
- Obscured, pointing upwards.
- Obscured, pointing downwards.
- Obscured, lying in the direction of separation.
- Not covered, lying in the direction of separation.

Critical for manufacturability are possibly the yellow colored areas *"not covered, in separating direction"*. The larger the required *Draft angle* is, the more areas are assigned to this category.

The three obscured surface types are shown in the same color because they define the surfaces next to insert or sliders. After hiding the part in the assembly tree, the surfaces can be selected individually, in order to make the division visible. You can also export each face as a file via the context menu.

3.10 Inner Curvature

For the analysis *"Inner Curvature"* a method has been implemented that marks missing fillets of inner edges. The threshold *Curvature radius* defines which minimum fillet (in mm) is required for inside edges of the analyzed parts. The missing fillet area is highlighted in the 3D-View and displayed as *Edge* in the results table. A fillet radius of 0mm will be displayed if the area is detected as hard edge without fillet. The individual found edges can be focused using the context menu of the result table, they can be exported and screen shots can be assigned per edge.

The implemented analysis is based on the networking of the loaded part. In the case of CAD data, you have the option of configuring the network quality in the user preferences, in order to improve the quality of the analysis.

3.11 Outer Curvature

Outside edges of casts should be rounded to make the part demoldable. In addition, missing rounding negatively affect the life of the part. In the *Outer Curvature* tab, a function is available that detects non-rounded outer edges and marks them in the 3D-View.

The threshold *Curvature radius* defines which minimum rounding (in mm) is required for outside edges of the analyzed part. The area of the missing rounding is highlighted in the 3D-View and displayed in the result table as an *Edge*. A rounding radius of 0mm will be displayed if the area is detected as hard edge without rounding. The individual found edges can be focused using the context menu of the result table, they can be exported, and screenshots can be assigned per edge.

The implemented analysis is based on the triangulation of the loaded part. With CAD data, you have the option of configuring the network quality in the user preferences, in order to improve the quality of the analysis.

3.12 Reporting

The function *"Reporting"* is used to document analysis results. A report summarizes information about the part and the *veoCAST* results. The report is supplemented by screenshots and user comments.

3.12.1 Screenshot creation

You have two options for creating screenshots for the report: *"Tab Screenshots"* document the overall results of an analysis tab, e.g. an overview of all maxima spheres found. *"Result screenshots"* show a single problem site and are assigned to a specific analysis result, for example to evaluate a maximum sphere.

3.12.1.1 Creation and administration of rider screenshots

To create a tab screenshot for an analysis, go to the corresponding tab. Adjust the 3D-View in size, aspect ratio and perspective as desired for the screenshot. Activate in the *veoCAST* toolbar the function *"Create tab screenshot"*. A screenshot of the current 3D-View will be taken. You can assign a name and a comment. The comment appears in the report next to the screenshot. *"OK"* saves the screenshot with the entered metadata, and with *"Cancel"* you can reject the screenshot. You can assign any number of screenshots to each tab. An overview of all captured screenshots of the tab can be found by selecting the *"Show tab screenshots"* in the *veoCAST* toolbar. You can use the context menu *"Edit Screenshot"* to display individual screenshots and to subsequently change the name or comment. *"Delete Screenshot"* removes the entry from the list of tab screenshots.

- The first screenshot of the tab *Administration* appears on the title page of the report and should document which part the report refers to.

3.12.1.2 Creation and management of result screenshots

Result screenshots can be assigned via the context menu of calculation results with *"Create Screenshot"*. You can assign multiple screenshots per result, such as an overview image, a detail image, and a sectional view for a maxima sphere. It is recommended to hide all other results for result screenshots (*"Hide Selected"*). With the context menu *"Show Screenshots"* you open the administration window of the screenshots for the selected result, which is structured analogously to the administration of the tab screenshots.

3.12.2 Export a report

Use the *Create Report* function in the *veoCAST* toolbar to export a document. In the file menu, you can select the location and file format of the report. The following formats are available:

- Portable Document Format (pdf)
- Office Open XML Text (docx)
- Office Open XML Table (xlsx)
- Open Document Text (odt)
- Hypertext (html)

The created reports can be changed. For example, Adobe Acrobat Reader allows you to draw and mark details in screenshots using the comment function.

3.13 Quick Review

The function *"Quick check"* makes it possible to start several analysis with the push of one button. In the *veoCAST* toolbar activate the function *"Quick check"*. Then the analysis *Maxima*, *Minima* and *Isosurfaces* are executed. The analysis will NOT start if the respective analysis has already been performed by the user. In this case, to perform the quick check, the analysis must be reset.

Configuration of the parameters for the quick check

For the automatically performed analyzes, the values stored in the *User Defaults* are used. To customize it, select *File > Preferences > veoCAST* and configure the values. The changes can be applied without restarting the application.

Quick check isosurfaces

The *User Specifications* for *Isosurfaces* also contain the parameter *Step size*, which is not included in the analysis tab. This value defines, at which intervals after the preprocessing isosurfaces are created.

Chapter 4

Advanced topics

4.1 Preferences

The default settings of *Veo* are stored in the *User Defaults* and can be adjusted by the user. Open the *User Preferences* under *"File"*>*"Preferences"*>*"veoCAST"*.

Calculation directory

During the analysis of geometry data, various geometries are generated, e.g. undercuts or tool halves. Each of these results will be stored separately in the calculation directory. The storage location can be specified via the field *"Directory"*. By default this is set to a local directory of the current user.

Basic settings

The basic settings of the analyzes *Maxima*, *Minima* and *Isosurfaces* are set in the user preferences under *veoCAST*. New *veoCAST* workbenches start with these values. The basic settings will continue to be used in Quick Check.

Chapter 5

Weitere Gestaltungsrichtlinien

Die folgenden Gestaltungsrichtlinien beziehen sich auf das Verfahren **Druckguss**, lassen sich aber z.T. auf andere Gussverfahren übertragen.

5.1 Bohrungen und Durchbrüche

Beim **Druckgussverfahren** können Bohrungen vorgegossen oder fertiggegossen werden. Eine geringe Konizität ist ausreichend. Zur Erzeugung von Bohrungen wird im Werkzeug ein Kern eingesetzt. Der Kerne ist einer hohen Belastung ausgesetzt, es muss sichergestellt werden, dass er nicht verschoben oder verformt werden kann. Durchgehende Löcher sind zu bevorzugen, da sie eine beidseitige Führung der Kerne in der Druckgießform zulassen. Das direkte Anströmen eines Kernes ist zu verhindern, aus diesem Grund ist auch eine Positionierung der Bohrungskerne im Anschnittbereich zu vermeiden. Bei dünnen Bohrungen und damit dünnen Kernen sollte ggf. nur eine Körnung gegossen und die Bohrung mechanisch hergestellt werden. Die folgende Tabelle gibt Grenzwerte für eingegossener Bohrungen an. Quelle: 1:

Werkstoff	Min.Durchmesser	Max.Länge durchgehend	Max.Länge Sackloch	Aushebeschräge
Aluminium	2,5mm	5*d	3*d	1,5 Grad
Zink	0,8mm	8*d	4*d	1,0 Grad
Magnesium	2,0mm	5*d	3*d	1,5 Grad
Kupfer	4,0mm	3*d	2*d	2,5 Grad

Bei Druckgussteilen ist zu prüfen, ob durch Bohrungskerne kritische Einschnürungen erzeugen. Diese können durch Drosselung des Metallflusses zu einem fehlerhaften Guss führen (siehe 1). Einschnürungen lassen sich in *veoCAST* durch die Analysen Minima und Isoflächen aufdecken. ## Einfluss der Bauteilgröße Druckguss-Bauteile werden anhand ihrer Raumdiagonalen in Größenklassen eingeteilt. Die Raumdiagonale lässt sich mit *veoCAST* aus den Länge / Breite / Höhe-Werten im Reiter Abmaße ermitteln (Wurzel aus L$L+B$B+H*H). Benötigt wird die Hüllquader-Diagonale bei Bestimmung von:

- Bearbeitungszugaben für Druckgussstücke aus Leichtmetallen (Al- und Mg-Legierungen) nach DIN 1688 Teil 4
- Für Kupfer-, Zink-,Zinn- und Bleilegierungen (Schwermetalle) DIN 1687 Teil 4
- Allgemeintoleranzen für Längen und Dicken von Druckgussstücken aus Al-und Mg-Legierungen nach DIN EN ISO 8062

TODO: Hier die Tabellen der Normen einfügen

5.2 Sonstiges

Weitere Gestaltungsrichtlinien für das **Druckgussverfahren** (nach 1):

- Große ebene Flächen werden besser gefüllt, wenn sie nicht glatt ausgeführt werden, sondern eine Oberflächenstruktur erhalten.
- Auswerferaugen oder -Dome vorsehen, um das Bauteil aus der beweglichen Form zu drücken, ohne es zu beschädigen
- Schriftzeichen sollten möglichst als erhabene Schriften auf dem Bauteil (vertieft in die Form) eingearbeitet werden. Dadurch ist die Form gegenüber dem einströmenden Gießmetall am wenigsten empfindlich und sichert den Gravuren hohe Lebensdauer.

Chapter 6

Feedback and Support for *veoCAST*

veoCAST is constantly being developed by the *Teraport GmbH*. For continuous improvement of the application, we rely on your feedback.

Error messages

If errors occur during use, it is important to describe the process in a reproducible manner. Click on the given mail address in order to send an error report: support@teraport.de

Suggestions

If you have suggestions for changes or improvements, please contact us

support@teraport.de

Phone Hot-Line

You can also contact us by the phone. You can reach our hot-line under +49-89-651086-703

Chapter 7

Literature

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