

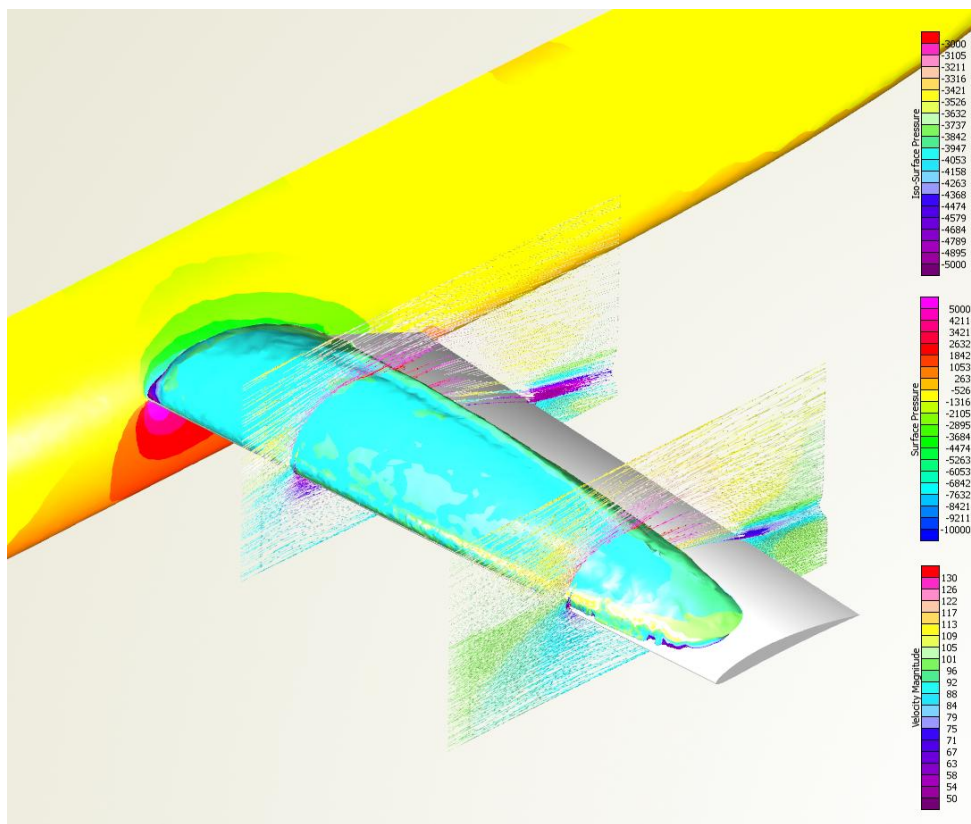
Overview

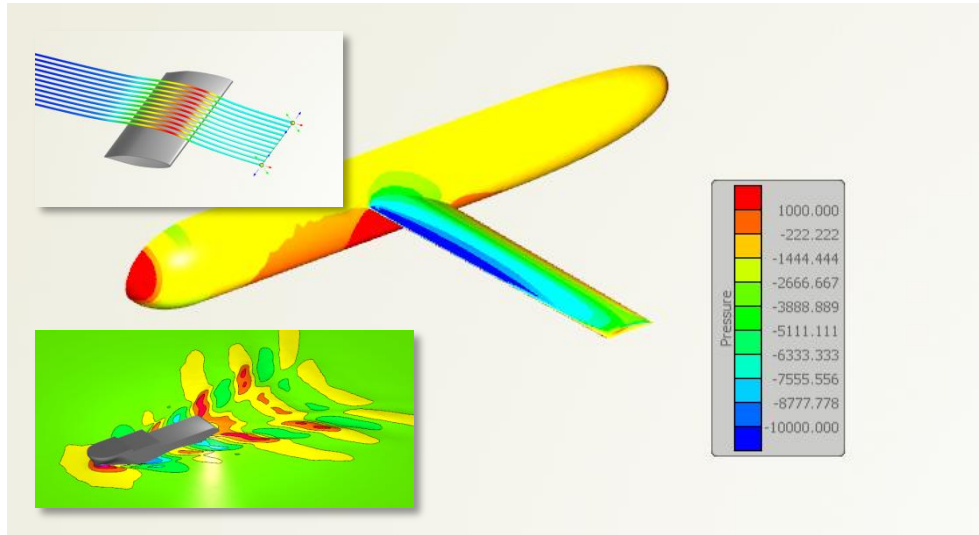


Postprocessing

CAESES provides functionality for the interactive postprocessing of simulation results. In this overview, the creation of typical visualizations such as contour plots and streamlines is illustrated.

If you have performed a CFD analysis in the wing tutorial, you can go through this tutorial interactively. Otherwise you can simply read through it to get an overview of the post-processing features available in CAESES.

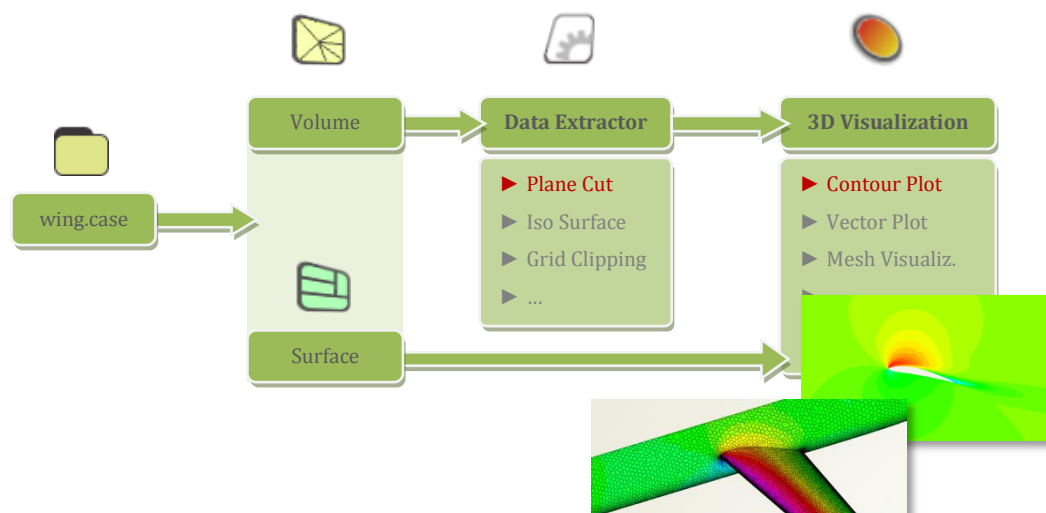




As a prerequisite, the tutorial “External Software” is recommended which explains basic functionality of the *software connector*, i.e. how to get the CFD results file from the external software into the graphical user interface of CAESES.

The Basic Idea

CAESES provides *data extractors* that are algorithms which extract certain data from a given data set, for instance performing a plane cut on a volume mesh. The output of such an operation can be visualized (e.g. display a contour plot on the extracted plane cut). If boundaries are already part of the flow solution, then you can directly apply a contour plot:



1

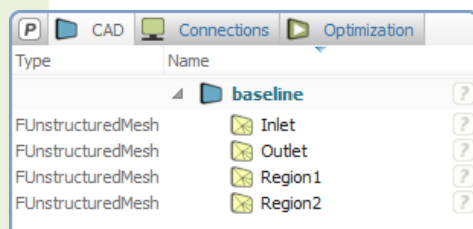
Simulation Data from Simple Import

Please note that this overview focuses on post-processing for CAD-CFD automation using the *software connector*, see step 2. However, simulation data can also be simply imported and visualized.

- Choose *menu > file > import* and choose from one of the supported formats.

✓ Currently supported formats are

- *EnSight Gold (.case)*,
- *OpenFOAM (.foam)*,
- *VTK*
- *Tecplot (ASCII)*



- Imported data are shown in the object tree (tab *CAD*).

✓ Remember: When importing result files via the file menu, their content is put into the *CAD* tree. In case a *software connector* gets used, everything will be put into the corresponding computation node (see steps 2 and 3).

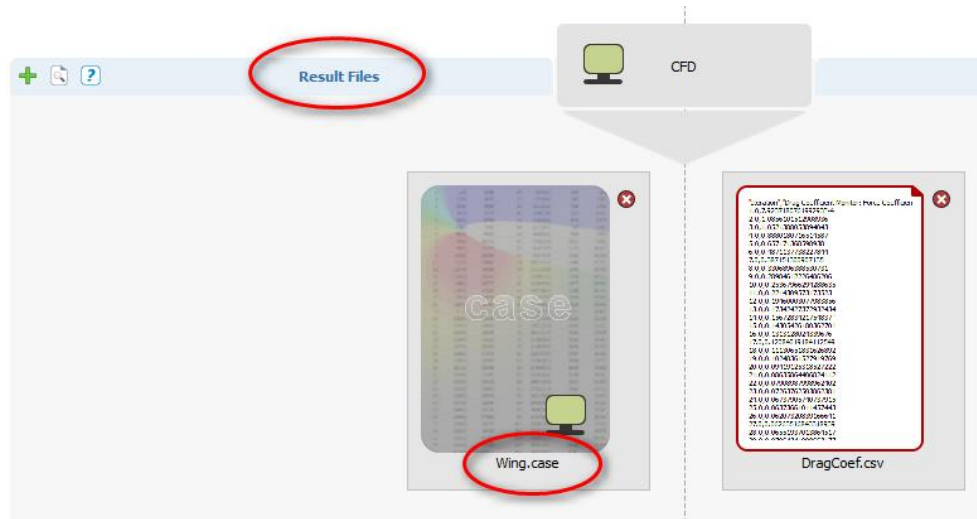
- You can now visualize data by following the next steps (jump to step 3).

2

Simulation Data from the Software Connector


In the context of CFD automation, data is provided by the *software connector* where CFD results are specified (see the previous tutorial).

- Add your CFD result data to the *result files* section of the software connector.





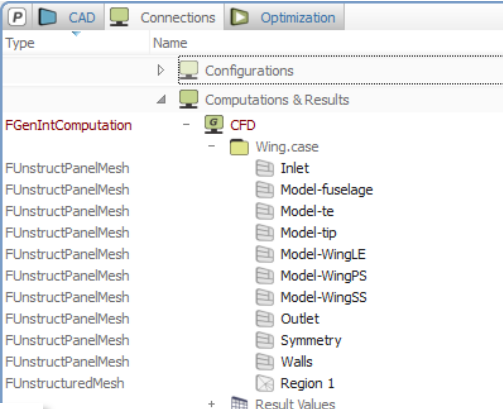
- After the computational run (either manually or automatically via *design engines*) the results are accessible in the object tree (tab *connections*, corresponding computation).

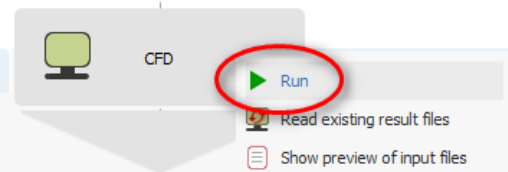
✓ A surface or boundary import (so-called *structured* or *unstructured panel mesh*) is indicated by this icon:



Volume data (*structured* or *unstructured mesh*) comes with different icons:

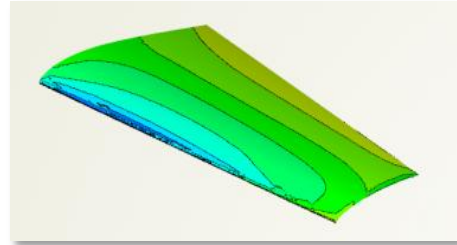




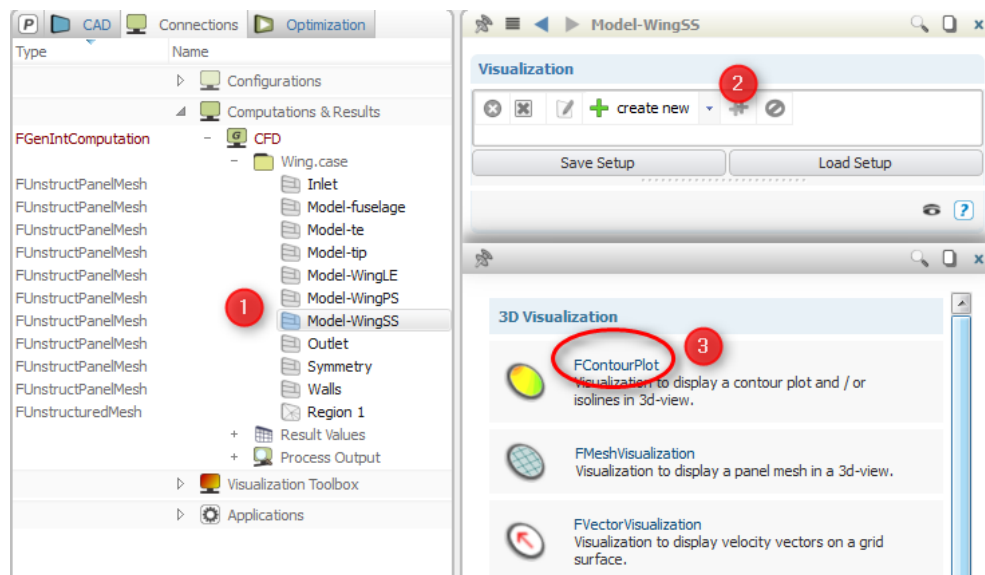
3

Contour Plot – Part 1

In order to illustrate the basic concept of post-processing in CAESES, we will visualize the pressure distribution on a wing surface with a *contour plot* where the flow data comes from one of the *EnSight Gold* result files.



- ▶ Select the boundary surface (1).
- ▶ Choose “create new” from the pull-down menu and click on “FContourPlot” (2, 3).



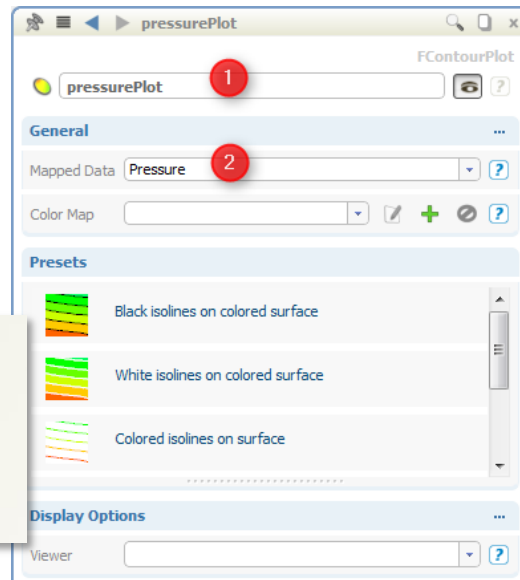
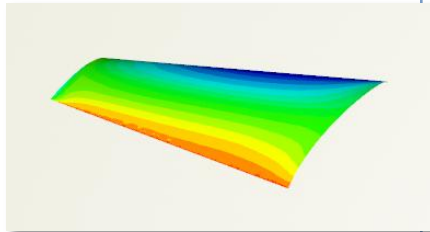
✓ For volume data, you need to extract a surface in order to generate a contour plot. This can be done for instance by using plane cuts (“FGridPlaneCut”, see step 7) or by extracting a grid surface (“FGridSurface”).

4

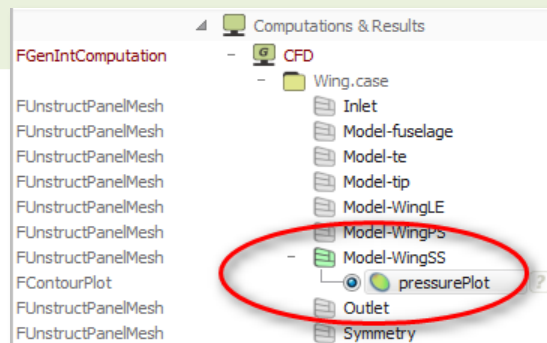
Contour Plot – Part 2

For the contour plot, you have to choose which field data you want to display. Also, setting a good name for each plot is recommended:

- Set a sensible name (1), which makes it easier at a later stage when having multiple visualizations.
- Select the *mapped data* (2). In our screenshot, we choose “Pressure”.



✓ Note that there is a new item now in the object tree, right below the previously selected object for which we started post-processing.

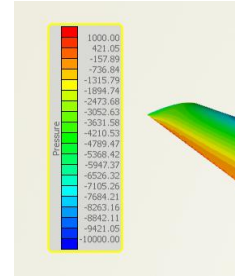


5

Contour Plot – Part 3

The software internally uses a default color map. Just to better understand the results, let's create a custom color map.

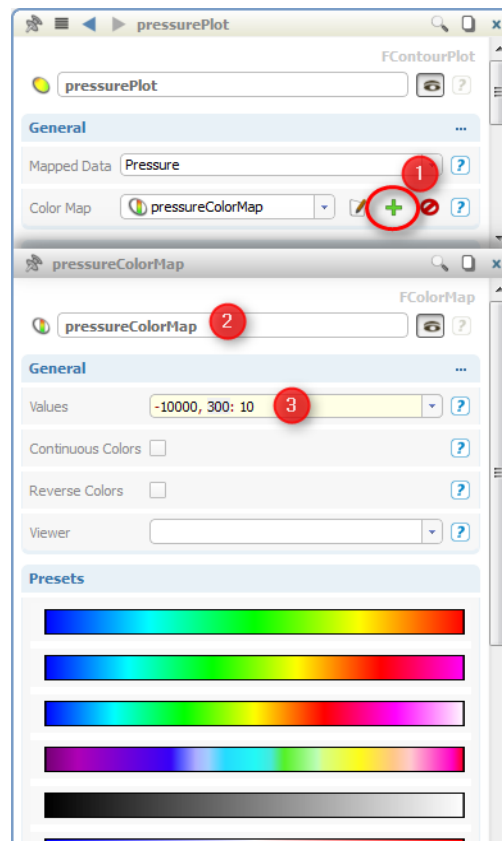
- ▶ Click on the "+" icon of attribute *color map* (1) which creates a new color map object.
- ▶ Set a name for the new color map (2).
- ▶ Adjust the *values* of the color map.



✓ See the documentation of "FSeries" for more information about number series. In the screenshot, the expression "-10000, 300: 10" is used which generates 10 equidistant numbers in the range [-10000, 300].

✓ The new color map item can also be used by other visualizations.

Important: If you now choose different *mapped data* in the object "pressurePlot" (e.g. "Velocity" instead of "Pressure"), you have to make sure that you still use the right color map!! Typically, you need to create another one for different field data.

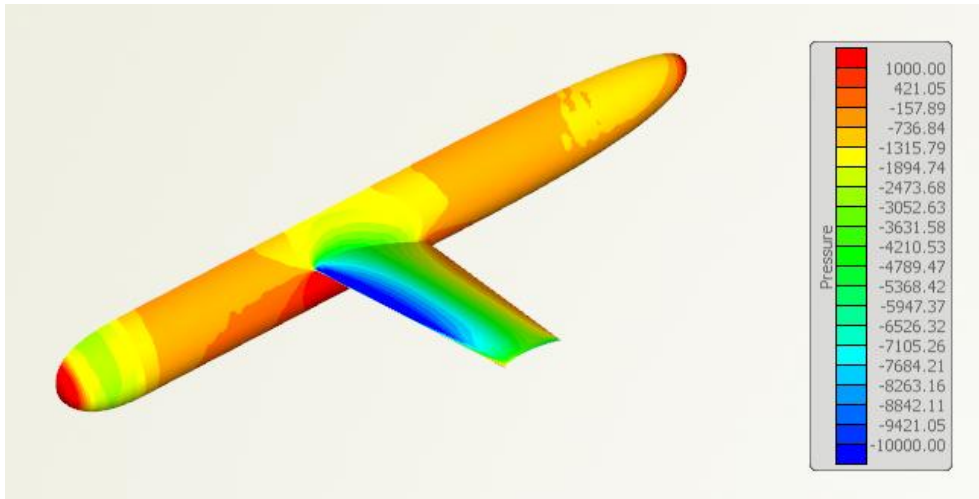


✓ If you select the color map object in the 3D view, it can be moved interactively.

6

Contour Plot – Part 4

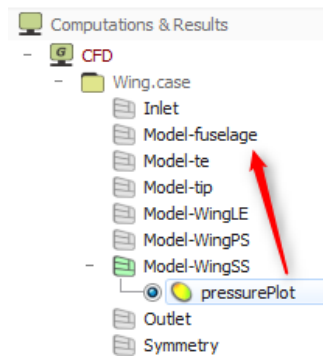
The contour plot is now configured for the wing's suction side. We can utilize this setup for other parts such as the pressure side and fuselage:



- Simply drag & drop the visualization to the target object.

✓ In our example, the item “pressurePlot” is dragged onto “Model-fuselage”.

Note that “Model-WingSS” and “Model-fuselage” are both boundary surfaces.



7

Grid Plane Cuts

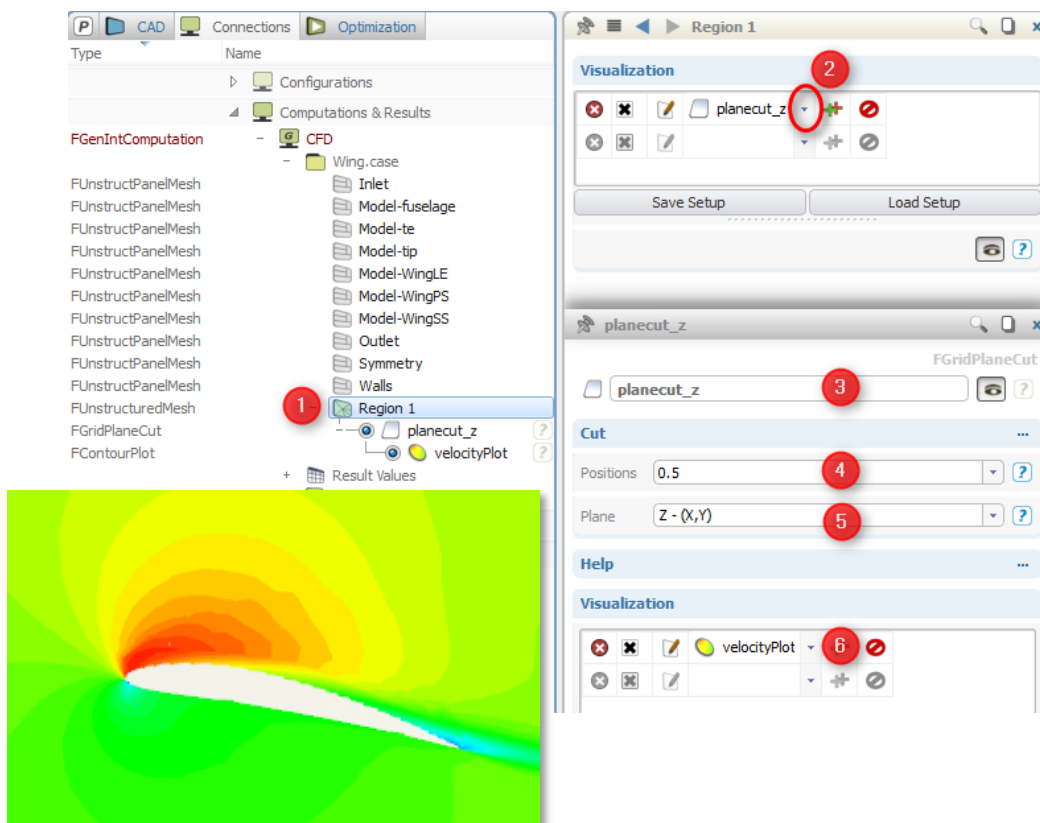
Creating a *grid plane cut* for e.g. a custom *contour plot* follows the same concept: We need to choose a region (i.e. volume data), select the plane cut as being the *data extractor* and, finally, visualize the extracted data by using a 3D visualization (such as a velocity contour plot or a vector visualization, see also step 11).



Data extractors are algorithms that perform a specific operation on a volume or surface mesh.

Keep in mind: They do not visualize anything!

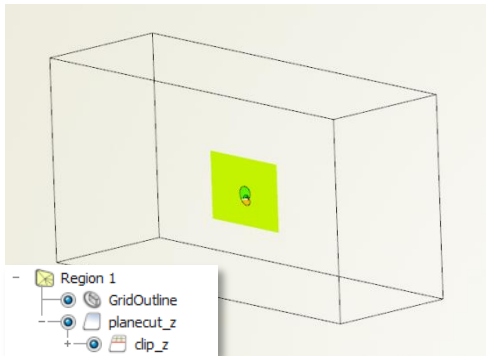
- ▶ Select the volume data (1).
- ▶ Choose “create new” from the pull down menu (2) and select “FGridPlaneCut”.
- ▶ Configure the plane cut, e.g. set a name, the principal plane, one or several positions (3-5).
- ▶ Create a contour plot again (6), see also step 3 of this overview tutorial.



8

Grid Plane Cuts – Including Clipping

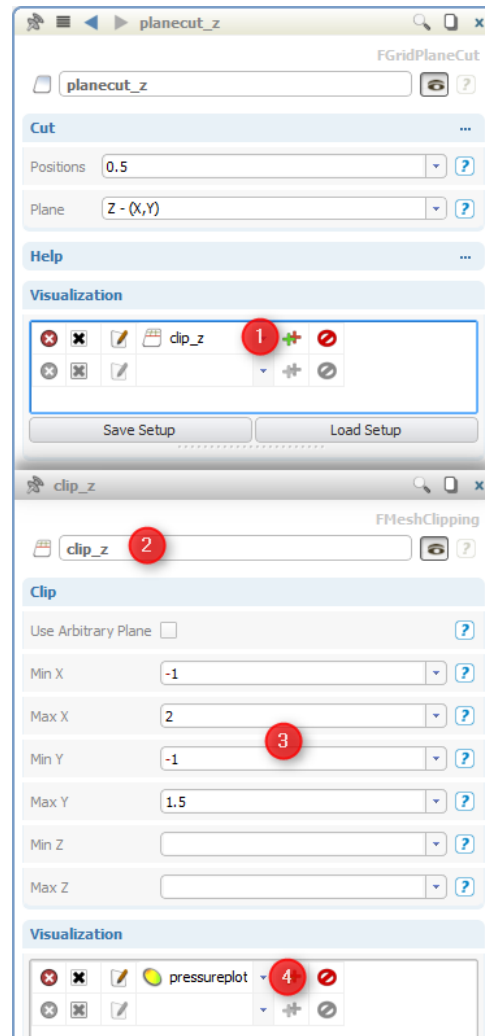
In some situations for large plane cuts you might want to reduce the range (see also the previous step for comparison):



- ▶ Select the volume data again.
- ▶ Choose “create new” from the pull-down menu and select “FGridPlaneCut”.
- ▶ Configure the plane cut (set a name, the principal plane, and one or several positions).

✓ You can also use a *grid outline* (choose “FGridOutline” from the menu) in order to see the bounding box of the volume data.

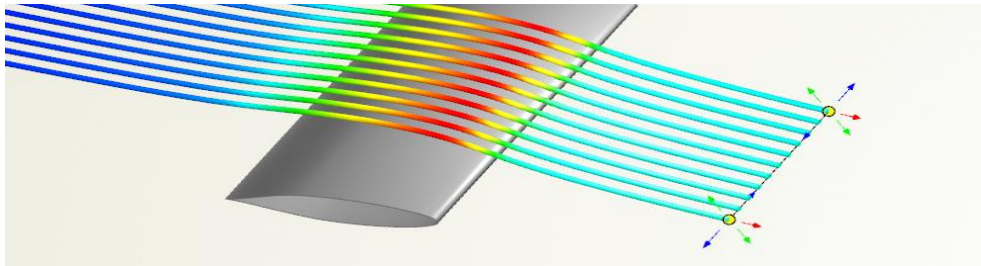
- ▶ For the *visualization* attribute of the plane cut object, choose “create new” and select “FMeshClipping” (1).
- ▶ Set a sensible name for the clipping (2).
- ▶ Set the minimum and maximum values for the x-, y- and z-axis (3).
- ▶ Finally, choose a contour plot of what you want to display on the clipped plane.



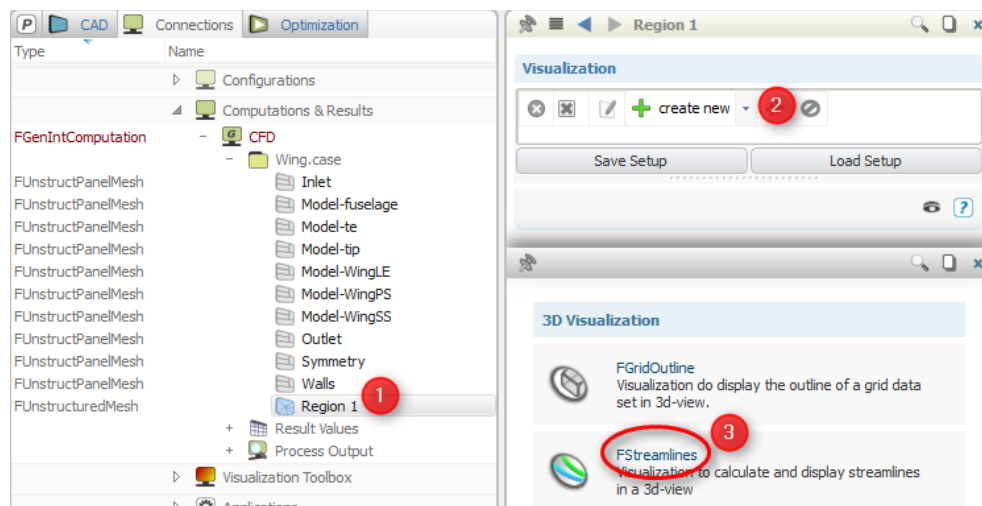
9

Streamlines – Part 1

Streamlines are used to visualize the flow trajectories and field quantities originating from user-defined seed points.



- ▶ Select the region i.e. volume data where you want to generate streamlines (1).
- ▶ Choose “create new” from the pull-down menu of your region (2).
- ▶ Click on “FStreamlines” (3).



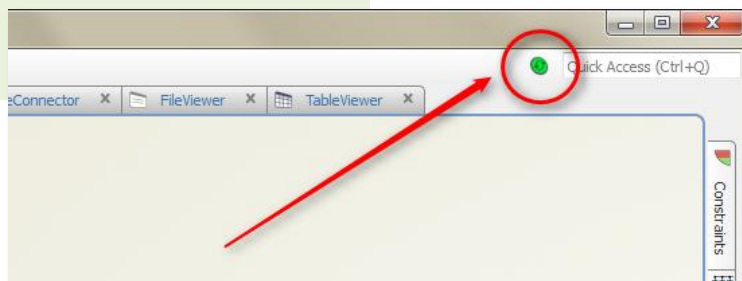
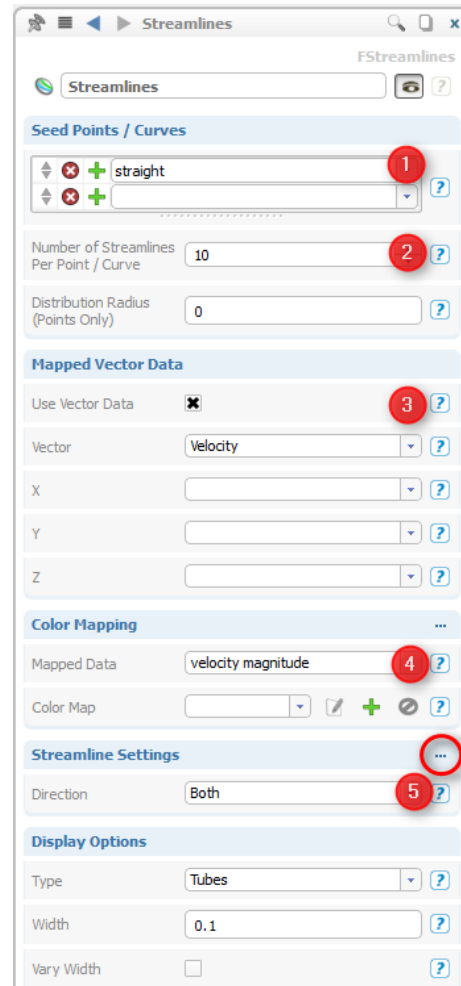
10

Streamlines – Part 2

In the second step, the streamlines settings are configured:

- Select *seed points/curves* from where the streamlines start (1). You can enter points and curves from your geometry model (tab *CAD*) or simple vector data such as “[0.1, 0.2, 0.3]”.
- Set the *number of streamlines* (2). For seed points, enter the *distribution radius* for generation of random samples.
- Choose a *vector* that will be shown (3). In the screenshot, this is “Velocity”.
- Choose *mapped data* that is used for color mapping of the streamlines (4). If needed, add a new color map for this purpose via the “+” button and configure it.
- Finetune the streamlines settings further by clicking on the header (“...”) (5).

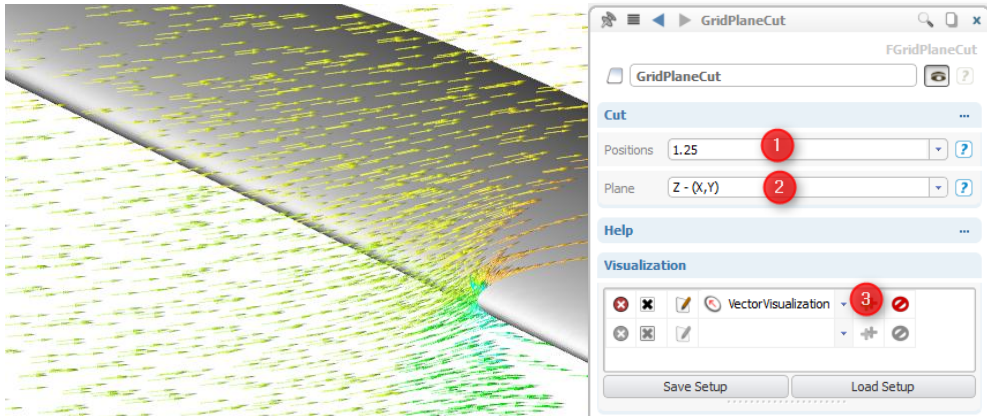
✓ Computing streamlines can be time-consuming. While changing the settings, you can switch off the auto-update. After settings are changed, switch it on again to generate the streamlines. Generally speaking, use this option for other visualizations if the update takes too much time.



11

Vector Visualization

Use a *vector visualization* in order to show flow directions which can be combined with a color mapping for e.g. pressure or velocity.

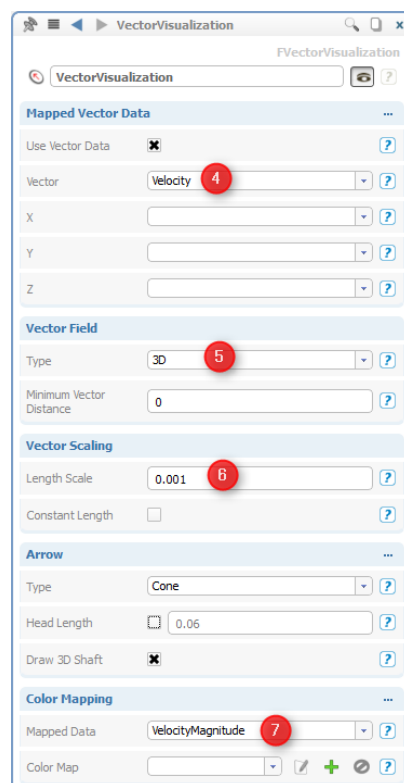


First, we extract plane data (i.e. surface data) from a volume by means of a *grid plane cut*:

- ▶ Select the region where you want to display vectors.
- ▶ Choose “create new” from the pull-down menu of your region and click on “FGridPlaneCut”.
- ▶ Set a position or multiple positions separated by commas (1).
- ▶ Set a principal plane for the cut (2).

Now, let’s visualize vectors on this *grid plane cut*:

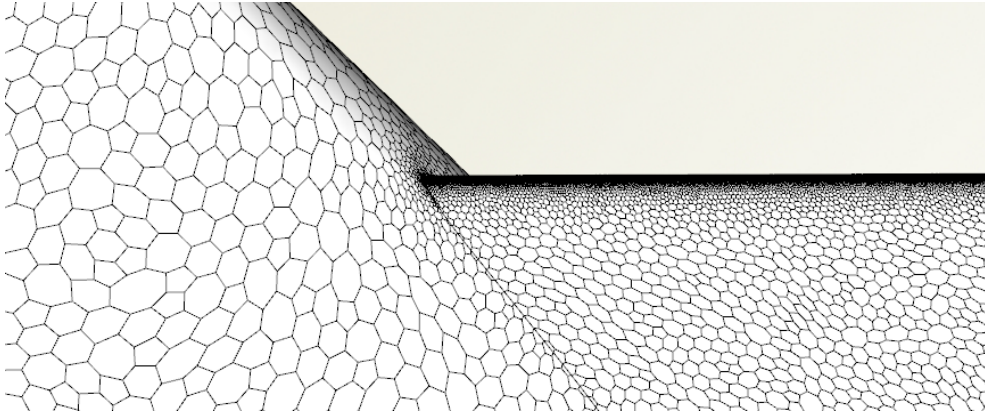
- ▶ In the visualization category of the plane cut, choose “FVectorVisualization” (3).
- ▶ Choose vector data to be displayed (4).
- ▶ Set the visualization type (5).
- ▶ Adjust the vector sizes (6).
- ▶ Choose which data is applied for color mapping of the vectors (7).



12

Mesh Visualization

If you want to simply display the mesh, then use a *mesh visualization* instead of, for instance, a *contour plot* as shown in step 3.

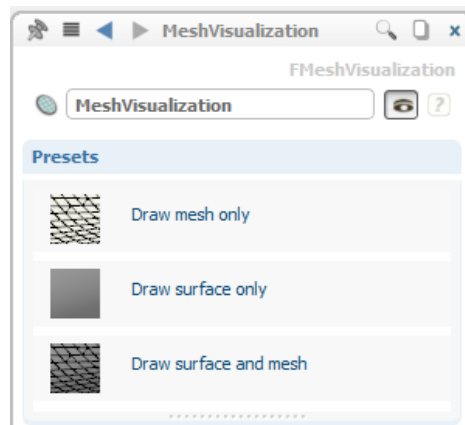


First, create a grid surface again in order to extract the mesh information:

- Select the boundaries where you want to display the mesh.

✓ You can select multiple boundaries (multi-selection) for assigning a new visualization at once.

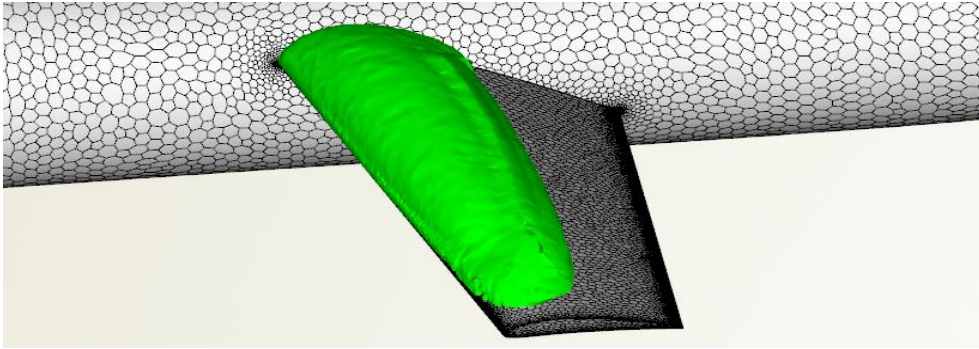
- Choose “create new” from the pull-down menu and click on “FMeshVisualization”.
- Configure this visualization further (presets, colors etc.)



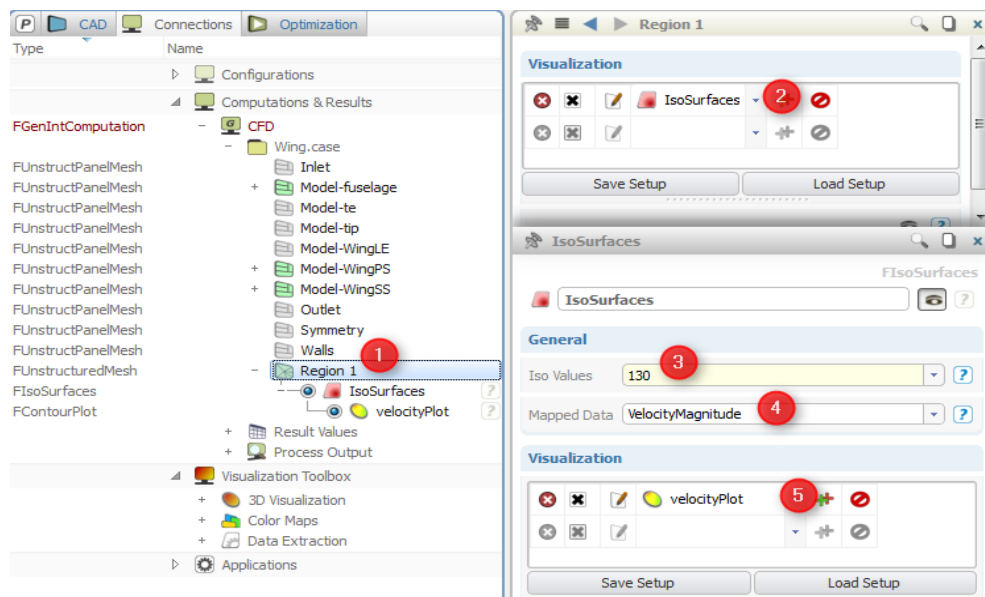
13

Iso-Surfaces

This type of visualization allows displaying a 3D surface where some quantity (velocity, pressure etc.) is constant.



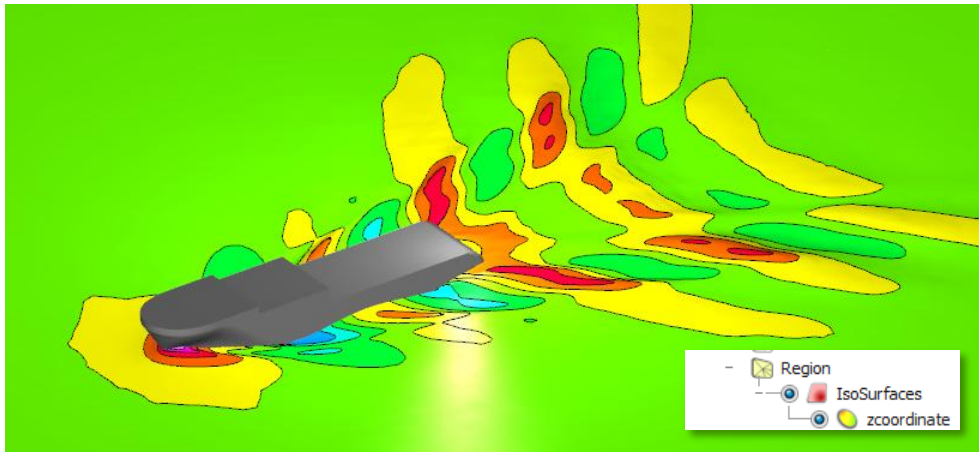
- Select the region where you want to display iso-surfaces (1).
- Choose “create new” from the pull-down menu of your region, click on “FIsoSurface” (2).
- Enter *iso values* (3) and choose from *mapped data* pull-down menu (4).
- Visualize the extracted surfaces by using a *contour plot* (e.g. create a new one).



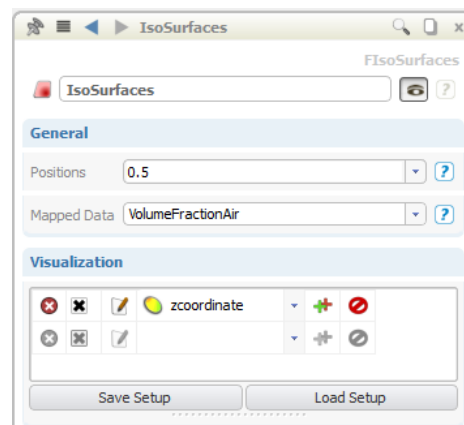
14

Iso-Surfaces – Wave Patterns

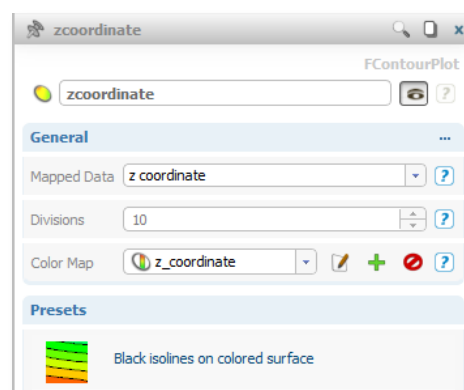
As another example for iso-surfaces, we can visualize the wave patterns from a CFD analysis of a ship hull.



- ▶ Select the region where you want to display wave patterns.
- ▶ Choose “create new” from the pull-down menu of your region.
- ▶ Click on “FIsoSurface”.
- ▶ For *iso values*, enter “0.5” (a volume fraction of 50% air / 50% water defines the free surface in typical maritime applications).
- ▶ For *mapped data*, choose the volume fraction of water or air from your CFD result data.



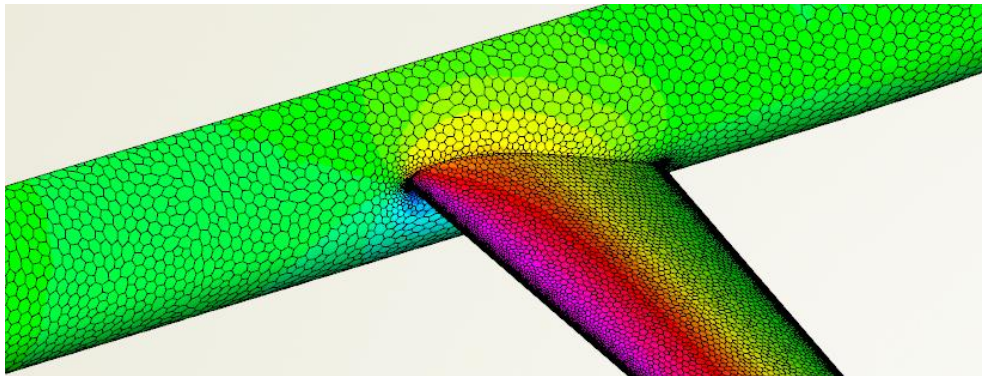
- ▶ Create a new *contour plot* (“FContourPlot”) via the visualization pull-down menu of the iso-surface (here: “zcoordinate”)
- ▶ Choose the z-coordinate as *mapped data*.
- ▶ Add a color map and use the presets.



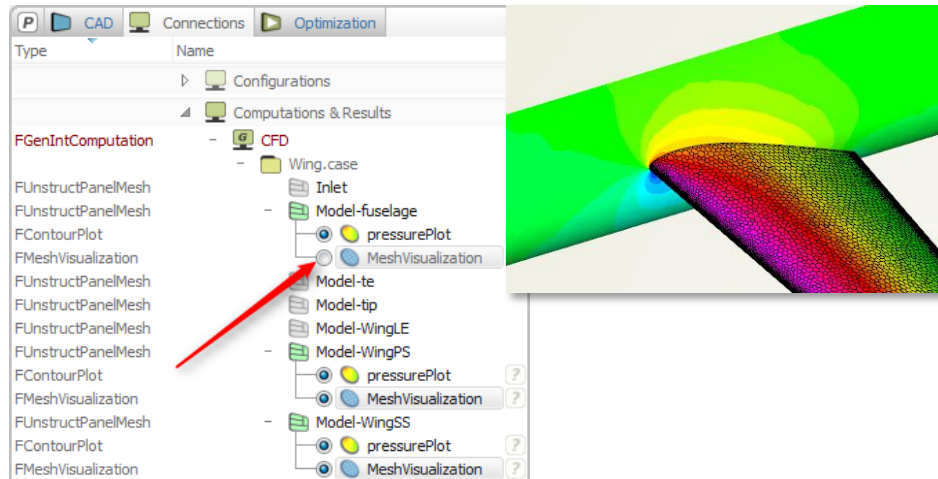
15

General Visibility: Local and Global

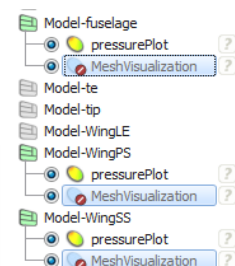
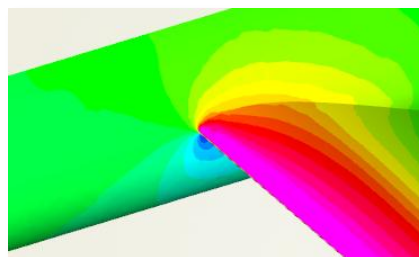
Back to our simplified aircraft: Visualizations and data extractors can be switched off and on in a local and in a global manner.



- If you want to deactivate a *data extractor* or *visualization* item for a specific part of your results (and the item is used for several regions or boundaries simultaneously), then click on the on/off-button next to the corresponding object. In our example, we deactivate the *mesh visualization* for the fuselage ("MeshVisualization" is also used for the wing):



- If you want to deactivate an item *globally*, then click directly on the icon! This will set the visualization invisible wherever it is used. In our example, the *mesh visualization* is deactivated for fuselage and wing.



16

Visualization Toolbox

In some situations it is handy to have quick access to a specific data extractor or visualization without browsing through the entire tree.

The node *visualization toolbox* in the object tree collects all created items for quick access:

- Simply select an item and configure it quickly. Note that this is a *global* change to the item i.e. changes are updated and displayed wherever the item is referenced.
- Click on the icon of an item in order to set it globally invisible (same functionality as shown in the previous step, second bullet point).

