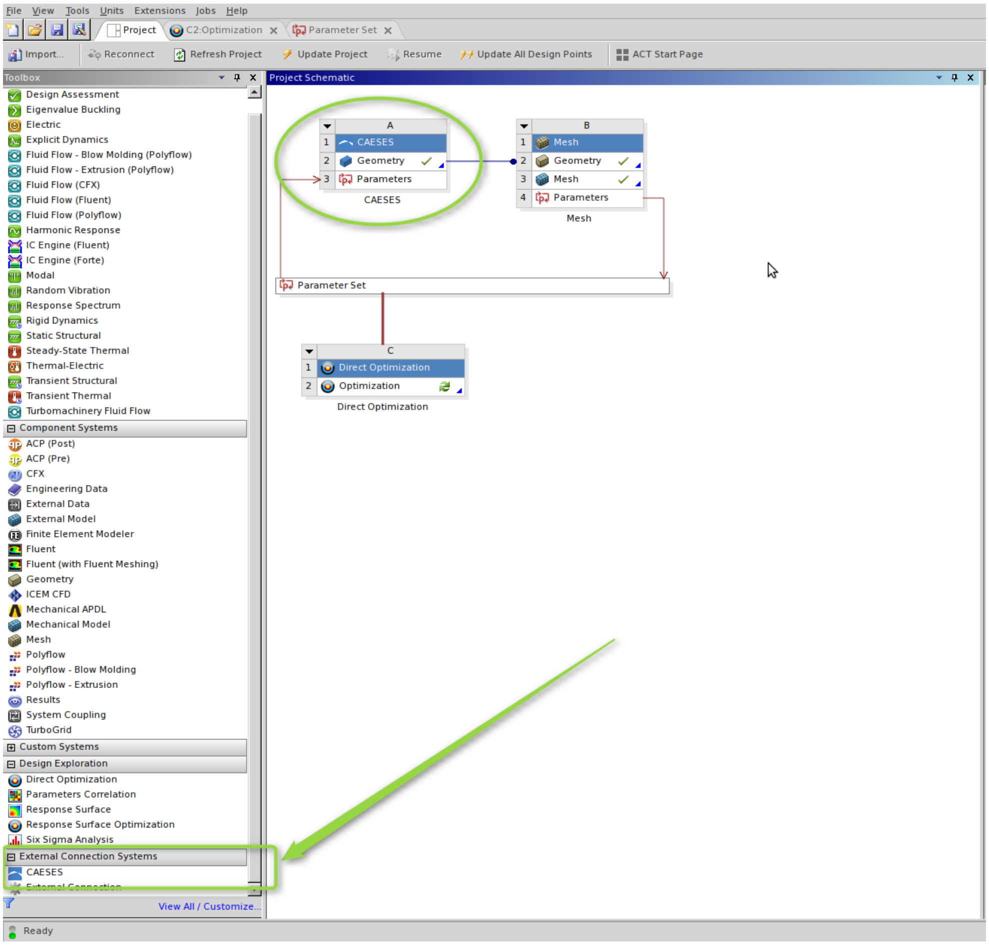
CAESES® External Connection Add-In for ANSYS® Workbench

This document describes how CAESES® can be used as an *External Connection Add-In* within the user interface of the ANSYS® Workbench. This integration allows you to generate new geometry variants directly from within the ANSYS® Workbench by changing the ANSYS® parameter set.



With this CAESES® Add-In, the full automation of the meshing and simulation process of new design candidates can be realized without any scripting.

This document also gives recommendations on the exported geometry formats to make use of automatic detection of so-called “Named Selections”.

- *Required CAESES® version: >= 4.1.3 (pro edition)*
- *Required ANSYS® version: >=17.2*

1

Making the CAESES® Add-In Available

In order to make CAESES® available as an Add-In in the user interface of the ANSYS® Workbench, we need to put a set of CAESES® files into the ANSYS® installation folder.

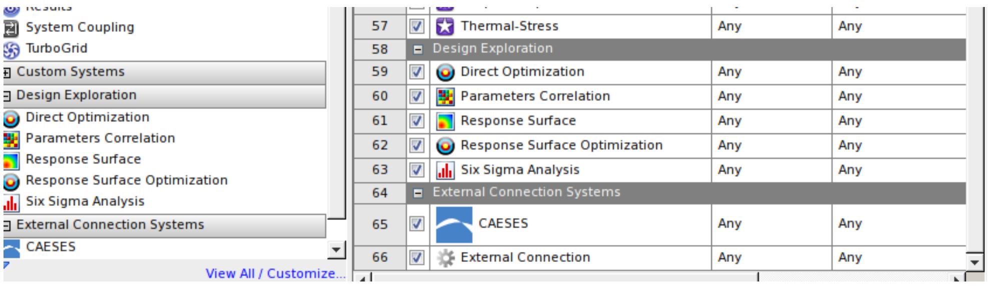
- ▶ Copy the entire folder *etc > ansys > SystemDefinitions*. It can be found in the installation directory of CAESES®.
- ▶ Paste the folder into the ANSYS installation directory (subfolder External Connection) so that you get the following structure:

ANSYS Installation/v172/Addins/ExternalConnection/SystemDefinitions/CAESES

✓ The folder "CAESES" contains the files that are needed.

If your directory already contains a folder that is called "SystemDefinitions", then paste only the folder "CAESES" into the "SystemDefinitions" folder.

- ▶ After restarting ANSYS®, the CAESES® Add-In is now visible on the left-hand side in the section "External Connections Systems" – as shown in the picture. In case it does not appear, choose *View > Toolbox Customization*, and activate the system "CAESES" in the corresponding widget:



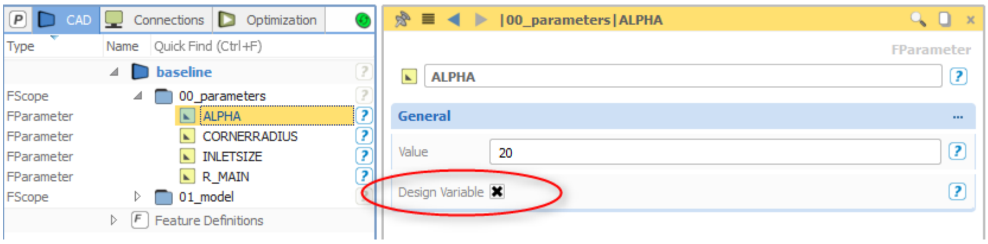
Preparing the CAESES® Project

Our motivation is to generate geometry variants from within the user interface of the ANSYS® Workbench. What actually happens?

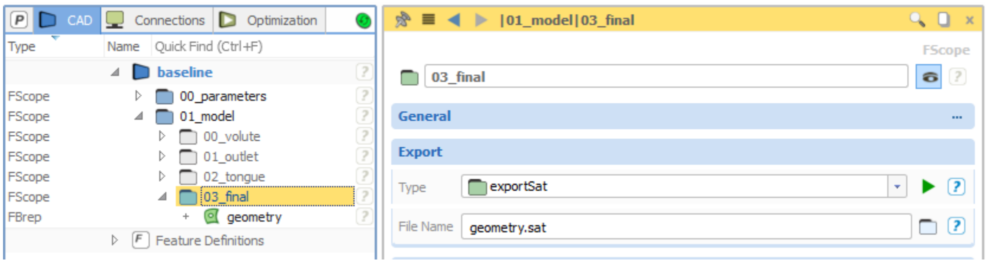
The CAESES® model will be updated in the background i.e. CAESES® is triggered in the batch mode, using the parameter input that you can enter in the ANSYS® Workbench. These values are transferred to the design variables of your CAESES® model, and a new geometry gets exported. The ANSYS® Workbench detects such a new geometry, and you can connect it to an ANSYS® mesh component, for instance.

All you need in your CAESES® project is a set of design variables as well as a configured geometry export:

- ▶ Make sure that you have at least one design variable in your CAESES® project.



- ▶ Create a scope and put the geometry objects that you want to export into this scope.
- ▶ Select the scope: Configure the export type and file name in the object editor.



✓ We recommend to use *.sat files. This file type gets supported by ANSYS® Workbench when it comes to "Named Selections" i.e. detection of boundary definitions in the geometry. This gets briefly explained in the appendix of this document.

3 Export the CAESES® Project as ANSYS® Setup

Each CAESES® project is an individual “configuration” of an external connection in the ANSYS® Workbench. CAESES® exports such a configuration file (*.xml) for your current project (*.fdb). This configuration can be read in again within the user interface of the ANSYS® Workbench.

- For an opened project in CAESES®, choose *file > export > ANSYS Setup*.
- Store it to a reference location on your PC.

This export stores a set of files into your reference location:

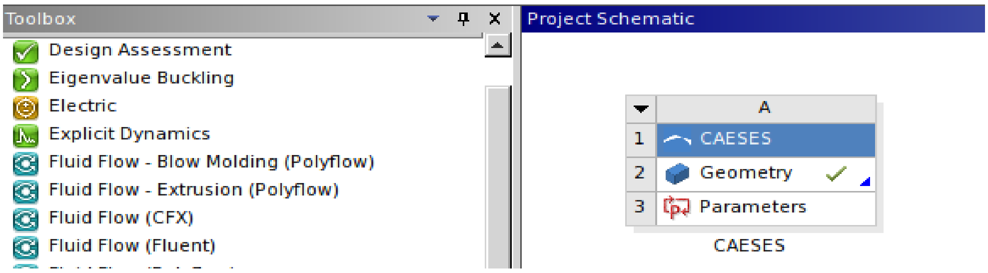
- *.fdb: A copy of your CAESES® project
- *.fsc: The script file for running CAESES® in batch mode with a set of parameters
- *.py: The python script that triggers CAESES® and takes care of the data exchange
- *.xml: A configuration file that tells ANSYS® about the parameters and the python file

✓ Note that this *reference location* is written into the exported ANSYS® files (*.xml, *.py). You should not manually change this location afterwards e.g. by copying the files from this location to another location.

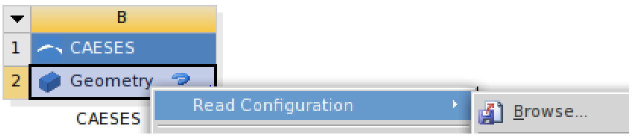
4 Load the Configuration into the ANSYS® Workbench

The export from the previous step writes a set of files from which we need the configuration file now:

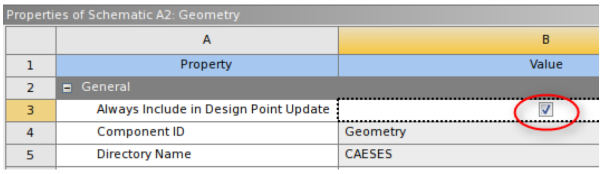
- At the beginning, drag & drop a CAESES® connection into the center widget of the ANSYS® Workbench so that a new container gets created.



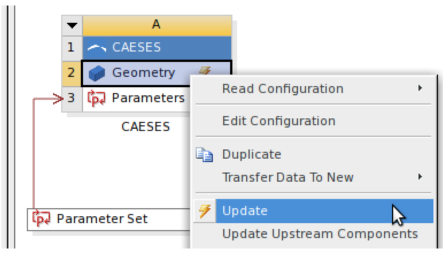
- In the context menu of the “Geometry” component, choose *read configuration* and select the file “<YourProjectName>_ANSYS_CONFIG.xml” from your reference location.



- Make sure that “Always Include in Design Point Update” is activated.



- Update the geometry which will run CAESES® in the background.

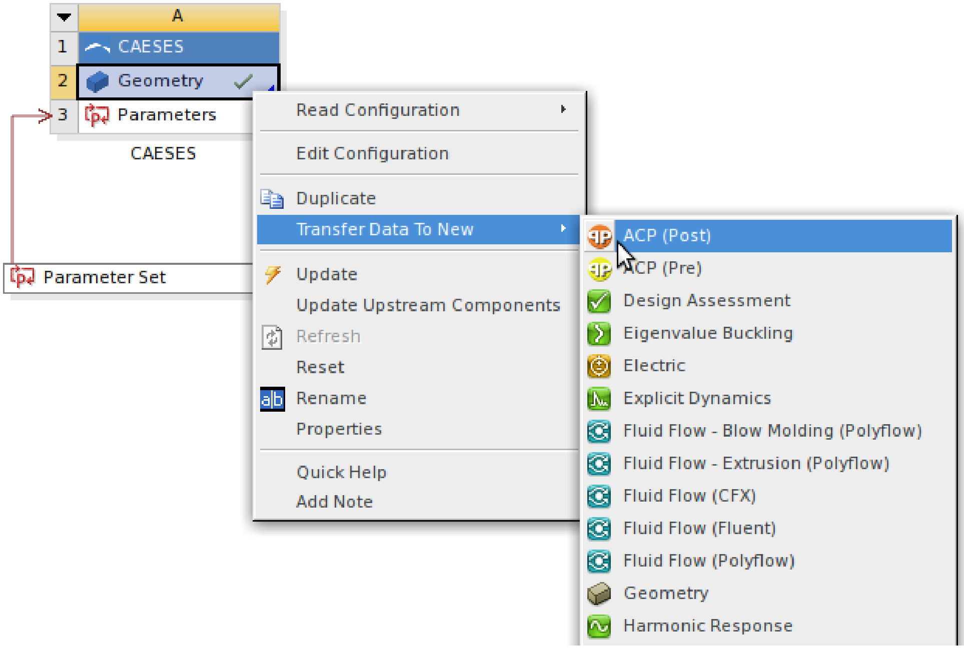


5

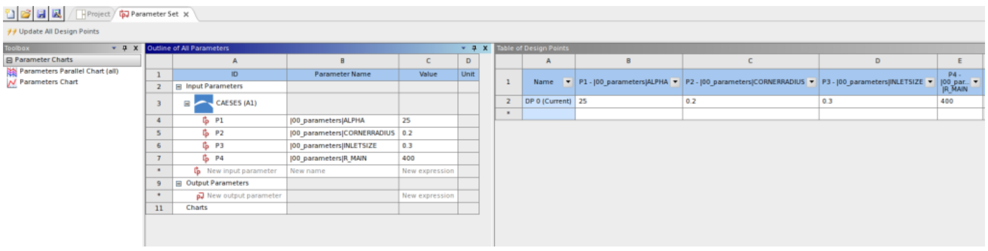
Continue Work in ANSYS® Workbench

Now your CAESES® geometry engine is ready and can be used. You can connect the component to the meshing, and you can change parameters:

- For the geometry component, choose “Transfer Data to New” in order to connect it to other containers of the ANSYS® Workbench.



- By using double-click on the “Parameter Set”, you can directly change the design variables of the CAESES® model. An update of the design point(s) triggers CAESES® in the background again.

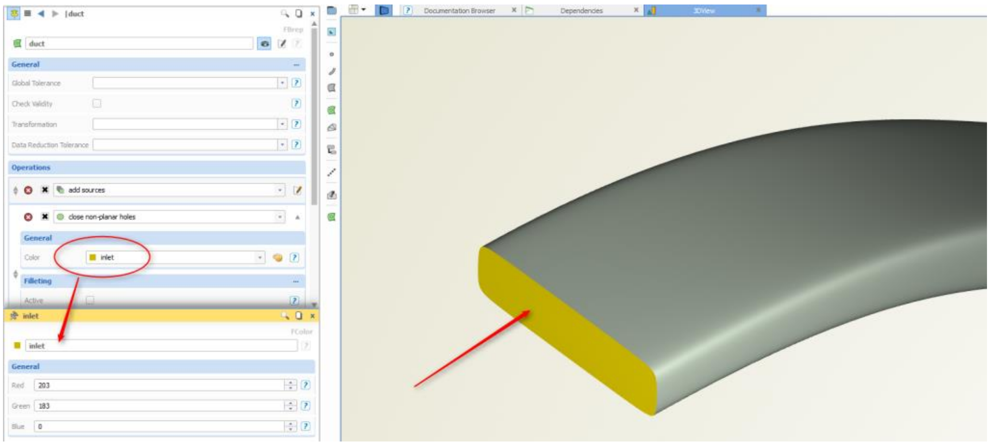


A1

Appendix: Boundary Definitions in CAESES®

The boundary definitions (inlet, outlet, wall etc.) can be already assigned in the CAESES® geometry. The information will be transferred through the export file and gets interpreted by the ANSYS® Workbench as "Named Selections" (see next page). This allows you to have a robust and fully-automated meshing even for models with topology changes (such as changing the number of blades for a turbine rotor etc.). This is what you need to consider in CAESES®:

- Assign the different colors according to the boundary definitions. You can use the default colors, or you can create own colors with more meaningful names. Names such as "inlet" or "wall" are easier to understand if you assign colors to several patches or geometries.



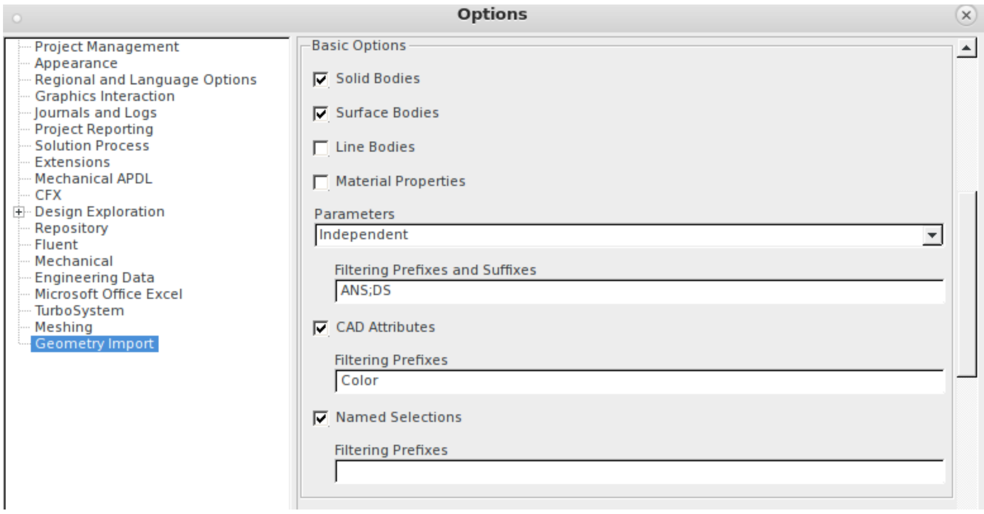
- As mentioned in step 2, choose the ACIS export (*.sat) as the export format for your geometry or scope.

A2

Appendix: Named Selections in the ANSYS® Workbench

We have to tell the ANSYS® Workbench that we want to take into account the color information that is given in the *.sat files.

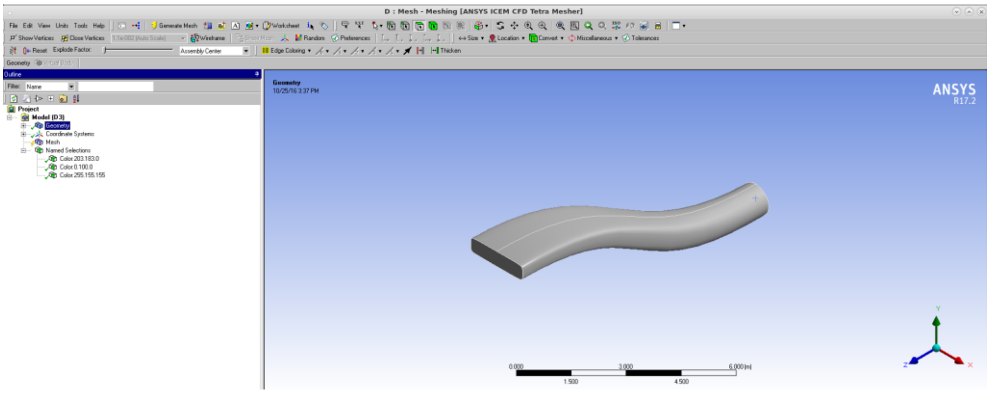
- Change the default options (*Tools > Options > Geometry Import*) in the ANSYS® Workbench:
  - Activate “CAD Attributes”
  - Enter *Color* as prefix to be filtered
  - Activate “Named Selections”
  - Leave the corresponding prefix field empty



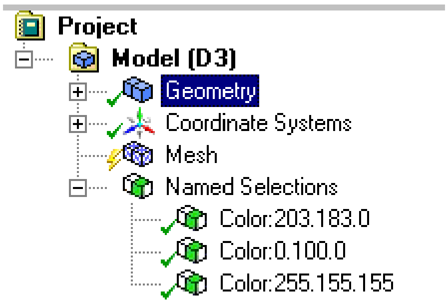
- Alternatively, change only the individual options of the CAESES® geometry component where you can use the same settings as described above:

Basic Geometry Options	
Solid Bodies	☑
Surface Bodies	☑
Line Bodies	☐
Parameters	Independent
Parameter Key	ANS;DS
Attributes	☑
Attribute Key	Color
Named Selections	☑
Named Selection Key	
Material Properties	☐

The following picture shows a result where the *Named Selections* are automatically detected by the ANSYS® mesher:



The following picture zooms in again into the widget on the left-hand side:



The geometry faces are grouped according to their color settings in CAESES®. Hence, you can continue with the details of your meshing procedure – all upcoming design variants will have the same *Named Selections* so that they can simply reuse your one-time meshing procedure.

This finally allows you to run fully-automated design studies and optimizations of any application with CAESES® and the ANSYS® Workbench with just a few clicks …