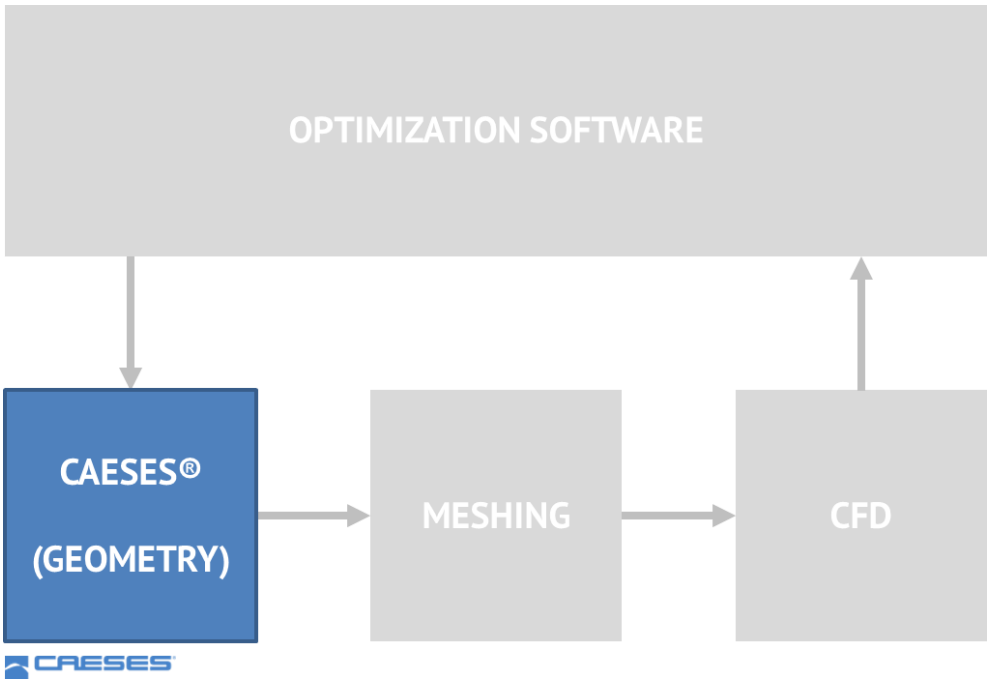
Using CAESES® in Batch Mode

This document describes how CAESES® can be run in batch mode. It allows you to embed CAESES® in workflows where other optimization tools exist already.

The design variables of a CAESES® model are typically changed by such external optimization software, and CAESES® generates and exports a new geometry.

The following picture shows an example of a workflow where CAESES® is only used for the purpose of geometry generation.

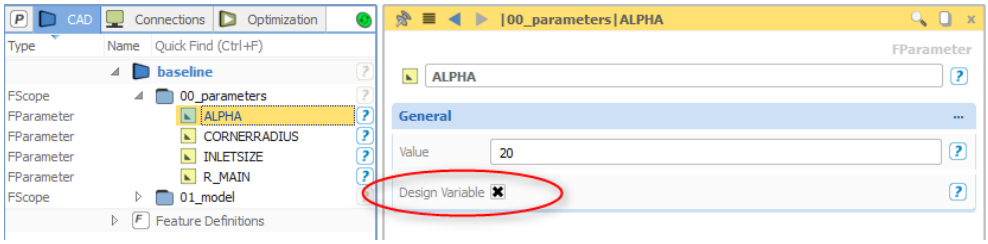


Note that the batch mode is available only for the pro edition of CAESES®. The document is based on version 4.1.3.

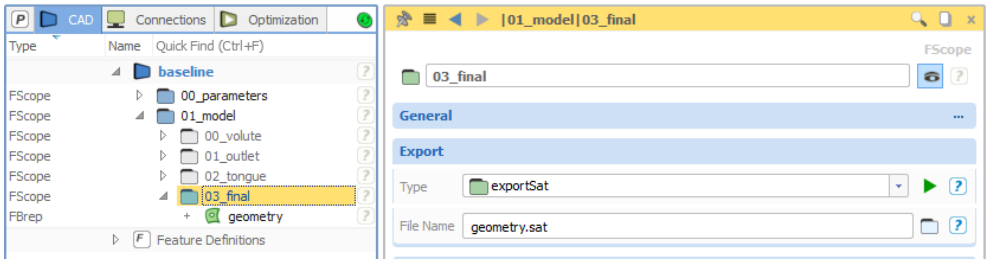
1 Preparing the CAESES® Project

In most situations, you only need to change a set of design variables of your geometry model. After the model update, you export a new design candidate by using a standard CAD format. This is your preparation for the project setup:

- Make sure that you have at least one design variable or parameter 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.



2 Generate a Script File

The CAESES® executable will be triggered together with a script file (*.fsc). This script file can be generated as follows:

- For an opened project in CAESES® as shown in the previous step, choose *file > export > FSC (CAESES Script)*.
  Make sure that nothing is selected during this operation, otherwise only the selection will be written into the file.
- Store it to a location on your PC, here is an example:

model.fsc

```
openProject("volutesymmetric.fdb")

// ------------------------------------------------------------------
// DESIGN VARIABLES
// ------------------------------------------------------------------

|00_parameters|ALPHA.setValue(20)
|00_parameters|CORNERRADIUS.setValue(12.5)
|00_parameters|INLETSIZE.setValue(75)
|00_parameters|R_MAIN.setValue(300)

// ------------------------------------------------------------------
// DESIGN VARIABLES END
// ------------------------------------------------------------------

|01_model|03_final|geometry.exportSAT("geometry.sat")

exit(false)
```

The generated file basically contains commands for the following tasks:

- Open your CAESES® project
- Change the design variables by using a setValue() command
- Export the new geometry
- Close the project

✓ Basically, you can use any object command in your script. For instance, you could run a persistent feature object by calling a command “myFeature.run()”. Simply type these additional commands into your fsc-script.

Running CAESES® in Batch Mode

We have the model setup (*.fdb) and a script file (*.fsc). As an example, we name the script file “model.fsc”. With these two files, we can run CAESES® in the batch mode. Use the following command:

- `C:\Program Files (x86)\FRIENDSHIP-SYSTEMS\CAESES\bin\win64\CAESES_crt model.fsc`

If you put this into a bat-file (Windows) then it will probably look like this:

- `”C:\Program Files (x86)\FRIENDSHIP-SYSTEMS\CAESES\bin\win64\CAESES_crt” model.fsc`

The quotation marks avoid problems with the blanks in a path. If you have another CAESES® installation directory, then choose it accordingly.

For Linux, there is also a crt-version in the CAESES® installation directory which can be used in the same way. If you need a separate script in Linux, an example script might be:

```
#!/bin/bash -x
INSTALLDIR/CAESES_crt model.fsc
```