

# Integrations



## Coupling of CAESES and XFlow

In this tutorial you will be guided step-by-step in the coupling process of CAESES and *XFlow*.

There is a [video available](#) which shows an XFlow integration by means of a high-lift airfoil.

You will learn how to set up the *Software Connector*, using the *Time Table Viewer* and *Time Series Viewer* and how to set up a variation. Before starting this tutorial please go through the tutorials *External Software* and *Jumper Integration*. These two tutorials show the basics of the software connector.



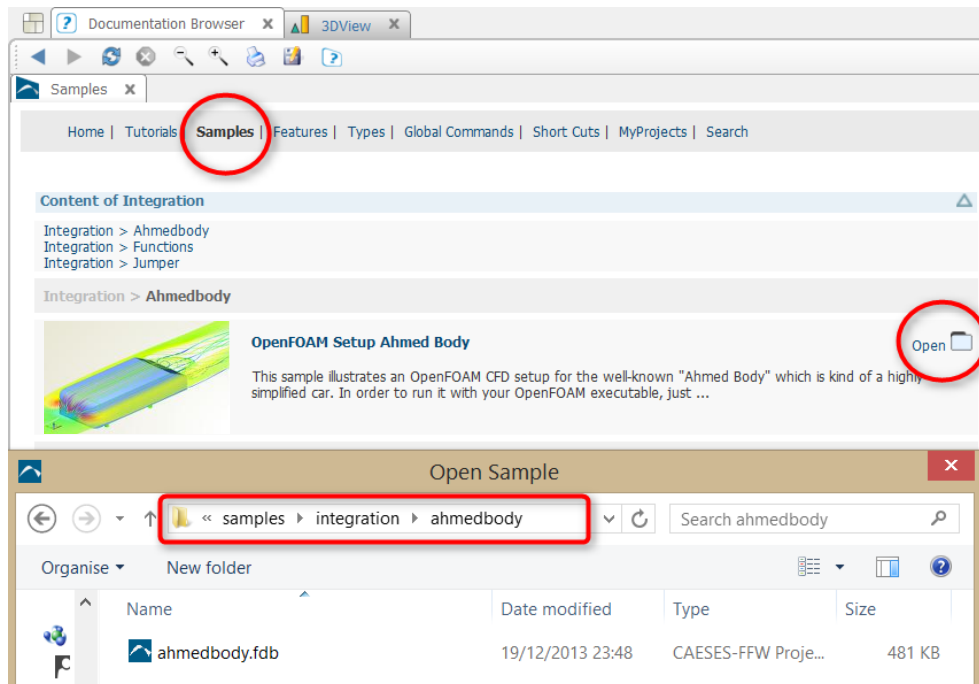
When you have finished this tutorial you'll be able to create your integration for CAESES and XFlow. Then you can start to vary and optimize your specific shape.

1

## Let's get started

In this step we will open the AhmedBody sample and use this project file for our XFlow integration.

- Choose *file > open sample* folder *integration > ahmedbody > ahmedbody.fdb*.
- Save a copy of this project via *file > save project* so that we do not modify the original project file.

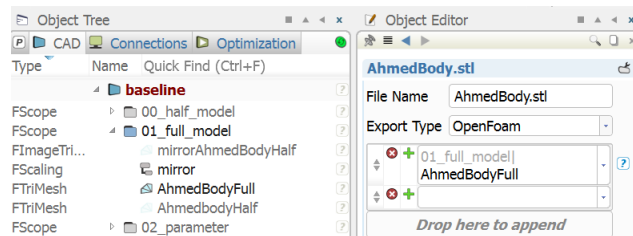


## 2

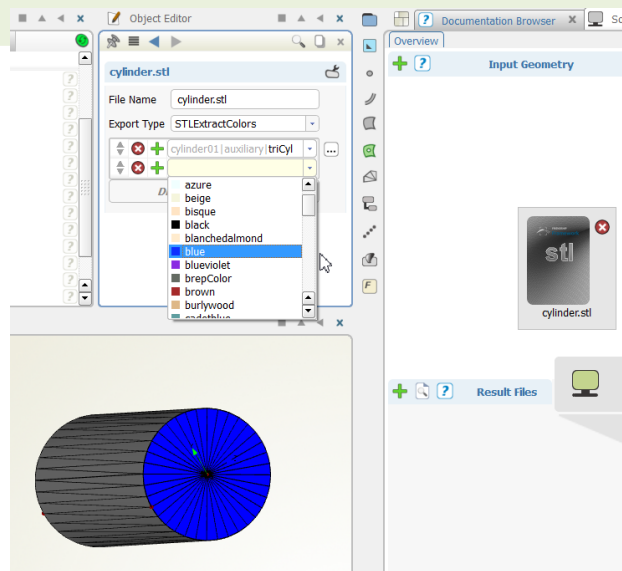
## Create a Software Connector

We will create a *Software Connector* and set the input geometry.

- ▶ Create a *Software Connector* via *Connections > Software Connector*.
- ▶ Click on the *CAD* tab of the object tree to access the geometry.
- ▶ Drag and drop the trimesh “|01\_full\_model|AhmedBodyFull” to the *Input Geometry* field.
- ▶ Choose the desired STL export type “OpenFOAM”.
- ▶ Set the file name to “AhmedBody.stl”.



✓ In this tutorial we are using the *OpenFOAM* STL export, but for internal flow problems you need to reverse the normals of some faces. We can do that by assigning different colors to faces and using the *STLExtractColors* export. Then you need to define the color of a face that should be reversed. Just add the color as source to the input geometry file of the Software connector. See picture below. The normals of the blue colored face will be reversed (pointing inside) after the export.



## 3

## Set the Input Files

We will import all of the necessary files for the coupling of CAESES and XFlow.

- ▶ Click on the green “+” button in the top right edge of the *Software Connector* to add an input file or use the right mouse button in the *Input Files* field and choose *Add Input Files*
- ▶ Select the XFlow project file in your installation directory > “tutorials/08\_Integration/04\_XFLOW\_Integration\_AhmedBody.xfp”.



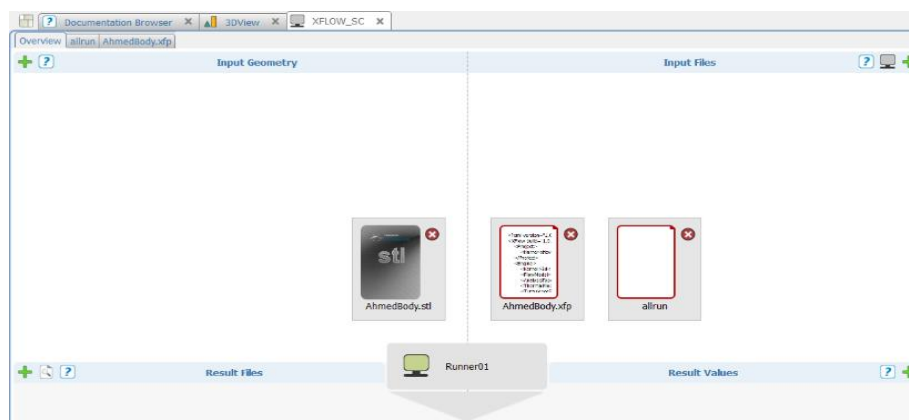
XFlow project files have the extension .xfp. It is an xml format file, which can be opened with an editor. Please also take a look at the XFlow Help, Chapter 8 for further information.

- ▶ Double click the file “04\_XFLOW\_Integration\_AhmedBody.xfp”. This file is now added to the project. It is a template and can be changed and linked to CAESES objects.
- ▶ Change the name to “AhmedBody.xfp”.



Reminder: Template files are not referenced to the original files on your hard disk. When we run the computation these files will be created. Template files are listed next to the overview tab. You can see the “Ahmedbody.xfp” being listed.

- ▶ Create a text file on your Desktop *right click > New > Text Document* and change the name to “allrun”.
- ▶ Double click the *Software Connector* widget and change the name to “XFLOW\_SC”.
- ▶ Add the “allrun” file.
- ▶ Double click the file “allrun” to add it to the project file.
- ▶ The picture below shows the *Software Connector* after these steps:

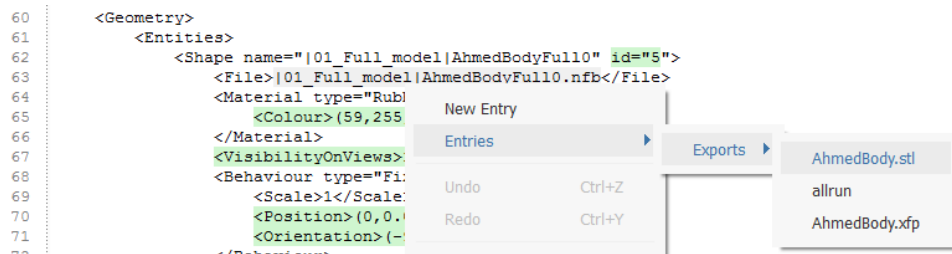


## 4

## Linking the Geometry to the XFlow Project File

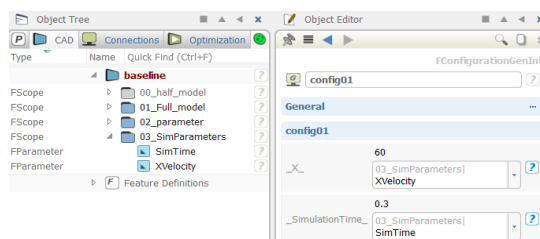
We will link the input geometry “AhmedBody.stl” with the XFlow project file “AhmedBody.xfp” and create parameters for simulation settings that we’ll be able to set and vary for XFlow with CAESES.

- ▶ Double click the template file “AhmedBody.xfp” or click on the tab “AhmedBody.xfp” next to *Overview* in the *Software Connector* widget.
- ▶ Go to line 63 and mark “|01\_Full\_model|AhmedBodyFull0.nfb”.
- ▶ Right click and select *Entries > Exports > AhmedBody.stl*. Now our STL file is linked to the XFlow project file.



As we want to be able to control the simulation time of our XFlow simulation and we want to create a parameter for the velocity in x-direction.

- ▶ Go to line 37 and mark the value “60”.
- ▶ Right click on the selection > *New Entry*; we create a new entry “\_X\_” in our configuration.
- ▶ Right click in the entry field of “\_X\_” and choose *create a Parameter*. Change the name to “XVelocity”.
- ▶ Go to line 85 and mark the value “0.3”, repeat the last two steps.
- ▶ Change the name of the parameter to “SimTime”.
- ▶ Select both parameters in the object tree and create a scope “03\_SimParameters”.



In the template editor you can use the CTRL + F to open the *find* dialog.

## 5

## Writing the “allrun” Script

We will write the allrun script to be able to start XFlow in batch mode. Information about XFlow running in batch mode can be found in the

XFlow Help Chapter 8.

- ▶ Select the software connector “XFLOW\_SC”.
- ▶ Open the template file “allrun”.
- ▶ Copy and paste the following lines into the “allrun” template file.

```
#!/bin/bash -x

#starts xflow in nogui mode and generates the binary files for the
simulation

/opt/xflow/92/xflow.sh AhmedBody.xfp -genbinaries

#switching into xflow simulation directory data=name of sim folder.
#This is necessary because generatedomain and engine needs to be
started in the simulation folder

cd data

#domain and engine

/opt/xflow/92/generateDomain3d AhmedBody.xfd

/opt/xflow/92/engine-3d AhmedBody.xfb

cd ..

#postprocessing data-export ensight or paraview

/opt/xflow/92/xflow.sh AhmedBody.xfp -exportdata=ensight -
exportfrom=3 -exportto=3 -exportdatatype=inst -
exportfields=vel,sp,cp
```



You need to change “/opt/xflow/92/xflow.sh” to your XFlow installation folder on your workstation.

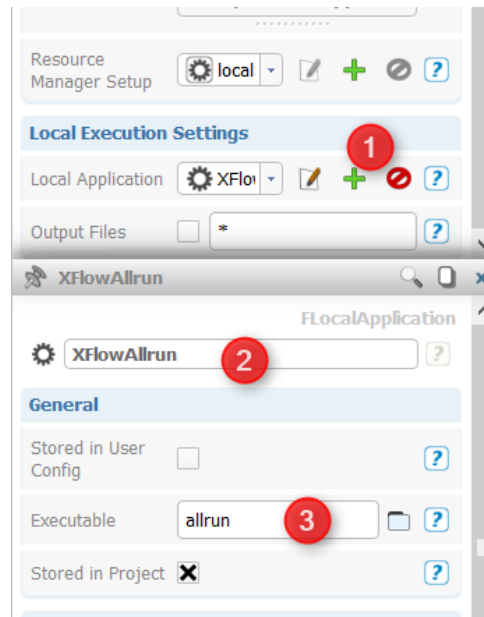
6

## Setting up the Computation

We set up the computation to link the previously created allrun script:

- ▶ Save your project (CTRL + S or *File > Save Project*).
- ▶ Click on *Overview* in the software connector.
- ▶ Click on “Runner01”.
- ▶ Change the name from “Runner01” to “XFLOW”.
- ▶ Click on the green “+” button next to *Local Application* to create a new local application (see the picture at number 1).
- ▶ Change the name to “XFlowAllrun” (2).
- ▶ Write “allrun” in the *Executable* box (3).
- ▶ Select *Stored in Project*.

Now the local application is saved in the project and always available and set.

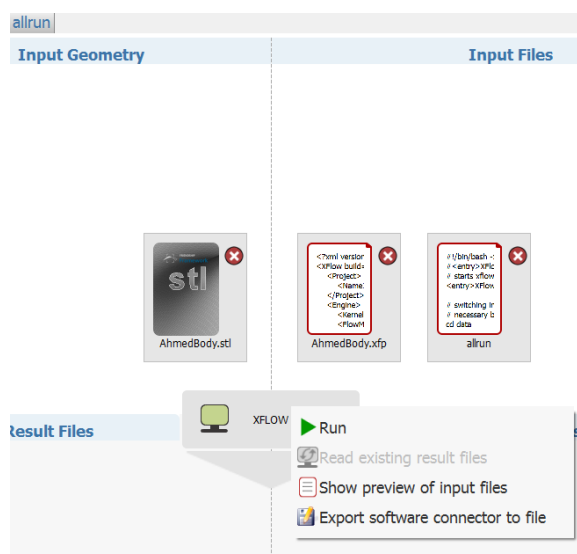


## 7

## Run the Simulation

We will run the XFlow simulation:

- ▶ Go to the *CAD* tab and change the parameter “SimTime” to “0.03”.
- ▶ Open the software connector “XFLOW\_SC”.
- ▶ Right click on the computation “XFLOW” and select *Run*.





## 8

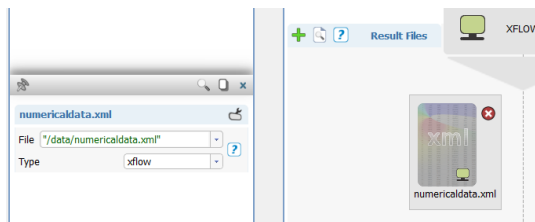
## Set and Read Result Files

After the XFlow simulation is finished we will add the file “numericaldata.xml” as a *Result File* and extract some values to calculate the drag.

- Go to the software connector “XFLOW\_SC” and right click in the *Result Files* field and select *Add Result Files*.

It will automatically open the folder of your computation.

- Open the folder “data” and select “numericaldata.xml”.
- Change *File* to “/data/numericaldata.xml”.
- Change the *Type* from “xflr” to “xflow”.



- Click on the computation “XFLOW” and click the button *read existing result files*.
- A time series viewer is opened and the console is showing this message:

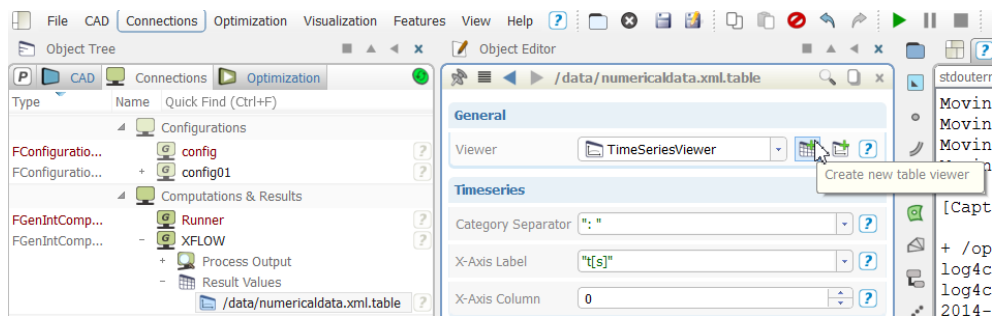
Console  
[16:34:56] \*\*\* INFO FXFlowParser : Importing 2222 time steps for 39 components

## 9

## Extracting Result Values from a Result Files

We will extract a value for the force.

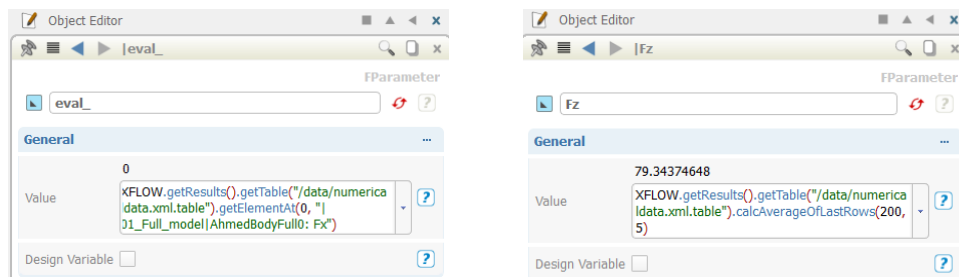
- Go to the tab *Connections* and select *Computations & Results* > *XFLOW* > *Result Files* > */data/numericaldata.xml*.
- Change the viewer by clicking on the table button (see screenshot). The results will now be shown in a table.



- Double click on the first row of column 5, with name "[01\_Full\_model|AhmedBodyFull0:Fz]".
- A parameter "eval\_" is created automatically in the *CAD* tab.

We want to calculate the average over the last 200 rows of that column.

- Delete "...getElementAT(...)" and type "calcAverageOfLastRows(200,5)". This command will calculate the average over the last 200 rows of column 5.



We need to use column 5 because the force in z-direction in CAESES is the force x-direction in XFlow.

- Change the name of the parameter to "Fz".



Reminder: You can find out the row and column indices by placing your mouse over one cell and they will be shown in a tooltip.

10

## Calculate the Drag Coefficient

We want to calculate the drag with the extracted force value from the last step. Here is the formula:

$$CD = \frac{Fz}{0.5 \cdot \rho \cdot v^2 \cdot A}$$

- ▶ Create a parameter via *CAD > Parameter* and change the name to “AreaFront”.
- ▶ We need to multiply “z\_height” with the “y\_width” parameter. You’ll find these two parameters in the scope “|01\_parameter|01\_side”.
- ▶ Create another parameter by pressing “F12”.

✓ Reminder: F12 creates the last created object again.

- ▶ Change the name to “CD” and create the formula below:

FParameter

↺
?

General ...

Value
0.30359894

?

Design Variable
☐
?

## 11

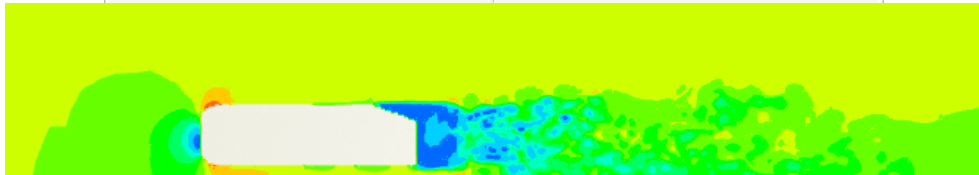
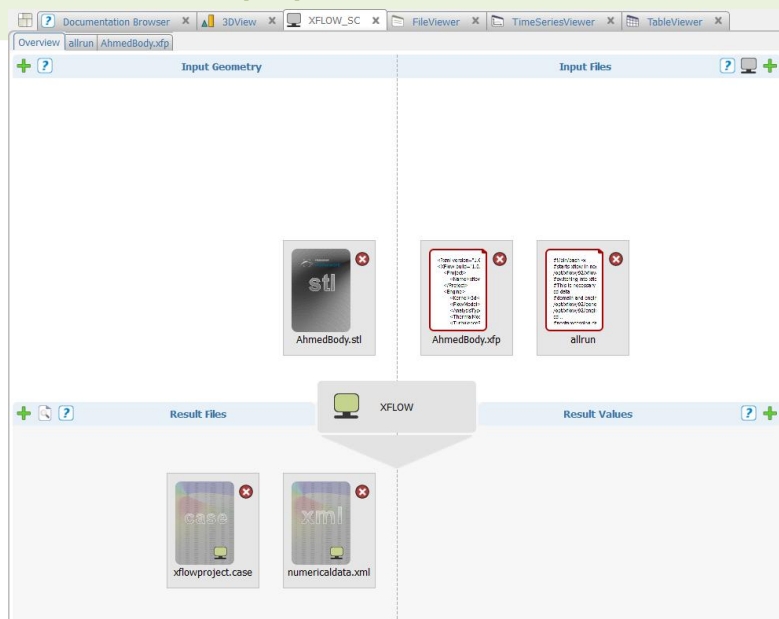
## Define Postprocessing Files

We will add the postprocessing data which we have defined to be created in our allrun script.

- Go to the software connector “XFLOW\_SC”, right click in the *Result Files* field and select *Add Result Files*.
- Navigate to folder “/data/exporteddata” and select “xflowproject.case”.
- Click on the computation “XFLOW” and click the button *read existing result files*.
- Go to the tab *Connections* and select *Computations & Results > XFLOW > domain*.

More information about postprocessing can be found in a separate tutorial.

✓ The postprocessing data format is *Ensignt Gold*. We defined this in the “allrun” script (see last command line) by setting the export argument to “Ensignt”. The format “paraview” will lead to *VTK* data files. See also the XFlow Help Chapter 8.



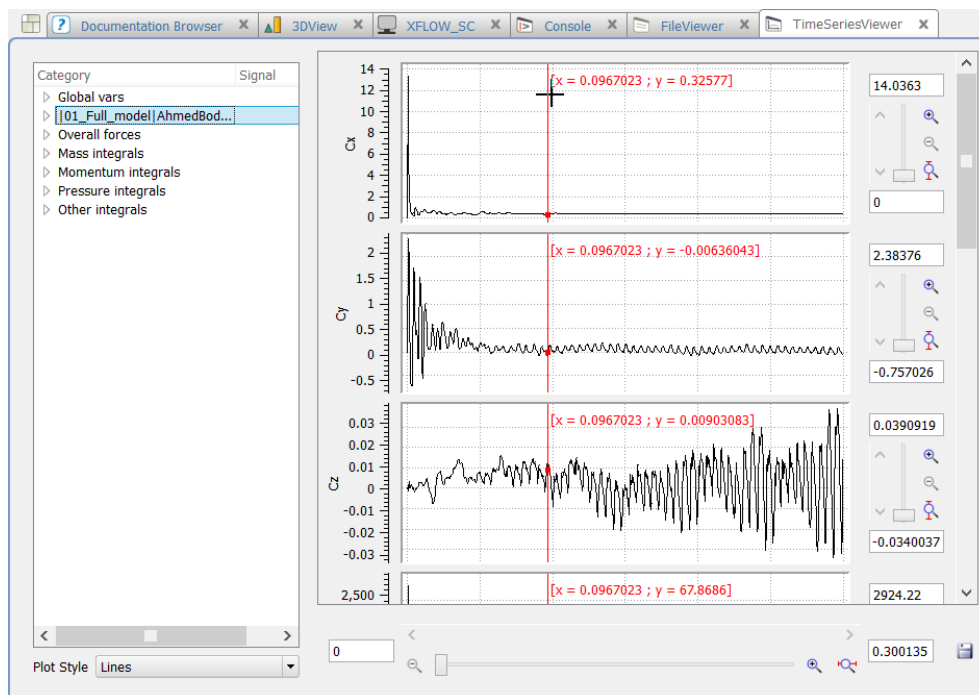
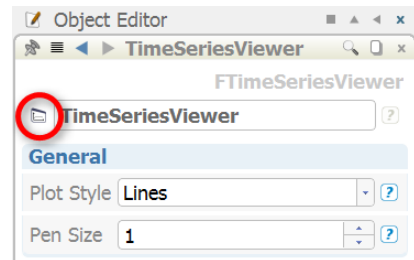
## 12

## Time Series Viewer

This step gives you some information about the *Time Series Viewer*.

Please also see the documentation in CAESES (double click on the *TimeSeriesViewer* tab (this selects the viewer), and then click on the *TimeSeriesViewer* symbol in the object editor. The documentation describes how to set up and use this viewer.

The time series viewer shows the function curves from the file “numericaldata.xml”. Each value is plotted in its own diagram. You can select a total category or specific diagrams that you want to visualize. Optionally, you can use this viewer in a *screenshot collection*.



## 13

**Outlook: Setting up a Variation**

Here is a short description of how to set up a simple variation of the ahmed body. We will use an *Ensemble Investigation* for the rear (slant) angle:

- ▶ Save your project again.
- ▶ Create an *Ensemble Investigation* design engine via *menu > optimization > ensemble investigation*.
- ▶ In the design engine, choose the design variable "slant\_angle".
- ▶ In order to set values for the angle, enter the series "0,5..45".
- ▶ Choose the parameter "Fz" and "CD" as *Evaluations*.
- ▶ Press the *Run* button to start the variation.

When all designs are finished you can see all designs in a result table. There, you can compare the designs along with their values.

Note: CAESES Free users can run variations only up to 4 design variants (5 in total).