

Coupling: OptiY – CATIA

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Interface **COM-Interface**
Experiment **OptiY\CAE Integration\CatiaAnsysIcemCfx\StaticMixer.opy**

Generate Script-File in CATIA

All working steps with CATIA let record into a script file by the menu *Tools/Macro*. The macro language must be VBScript. After the macro editor has started, do following steps in CATIA:

- Open the CATIA model.
- Select and change values of all model parameters
- Update the geometry
- Save the file to a given format e.g. IGES

After all steps are done, close the macro editor and open the script file with a text editor.

Setting in OptiY

Insert some of the source codes of the script file to the global COM variables in OptiY-Script-Editor.

Define the global COM variablen for OptiY and Catia:

```
Set Opt = CreateObject("OptiY.Application")  

Set CATIA = CreateObject("CATIA.Application")
```

Open the CATIA model and define global variables for further use. Please pay attention to the correct path of the experiment:

```
Set documents1 = CATIA.Documents  

Set partDocument1 = documents1.Open("C:\OptiY\CatiaAnsysIcemCfx\StaticMixer.CATPart")  

Set part1 = partDocument1.Part  

Set bodies1 = part1.Bodies  

Set body1 = bodies1.Item("MainBody")
```

All model parameters must be assigned to according variables. e.g. the model parameter *hybridShapeRotate1.Angle* of the part *Drehen.1* is assigned to the variable *in2angle*:

```
Set hybridShapes1 = body1.HybridShapes  

Set hybridShapeRotate1 = hybridShapes1.Item("Drehen.1")  

Set in2angle = hybridShapeRotate1.Angle
```

These defined variables have to be linked with nominal and tolerance variables in OptiY experiment. The scripting language of OptiY is set to VBScript. The Extern Script contains following codes to update geometry and save file as IGES:

```
part1.Update  

partDocument1.ExportData "C:\OptiY\CatiaAnsysIcemCfx\StaticMixer.igs", "igs"
```

Please pay attention to the correct path of the experiment.