

Coupling: OptiY – Abaqus

OptiY e.K., Germany
www.optiy.de

Interface **ASCII-File**
Experiment **OptiY\CAE Integration\ABAQUS\TestPython.opy**

Python-Script in Abaqus generieren

The Abaqus model must be existed as python script. The working procedures with GUI let keep as macro-file (here: Beam.py), from which the python script can be gotten and edited. All required parameters have to be defined at beginning:

```
# Parameters
Length = 197.84
Width = 30.324
High = 25.27
Pressure = 0.5
Modulus = 209.0E3
```

All numbers have to be replaced by according parameters. After simulation, the results being investigated should be gotten from the data base. First, the data base has to be opened:

```
# Open Data Base
myOdb = openOdb(path=jobName + '.odb')
assembly = myOdb.rootAssembly
```

Then, calculate the results (here: maxMises) and save it to an extern ASCII-File (here: Mises.txt):

```
# Write max. Mises to ASCII-File
outputFile = open('Mises.txt','w')
outputFile.write('Max. Mises = %f' %(maxMises))
outputFile.close()
```

At last, close the data base

```
# Close Data Base
myOdb.close()
```

Setting in OptiY

The generated python script is embedded as ASCII-Input-File in OptiY. Extern Script (DOS-Batch) performing simulation contains following codes:

```
abaqus cae noGUI=Beam.py
del Beam_Job.*
del abaqus.*
```