

### **Tutorial:**

# CAARC building-FSI simulation OpenFOAM-CoSimulation-Kratos

Ashish Shankar, Darekar

Chair of Structural Analysis

**Technical University of Munich** 

Munich, 25. July 2022

Email: ashishdarekar14@gmail.com







# Introduction

This tutorial shows how to use OpenFOAM as a fluid solver and Kratos as a structural solver to simulate the CAARC building-FSI using MPI. The CoSimIO tool is useful for inter-solver communication using "File" based communication, while the Kratos-CoSimulation application is useful for coupling.

One can refer the Master's thesis, "Development of a general OpenFOAM-Coupling-Adapter for the Kratos-CoSimulation Multiphysics solver," to address further FSI problems that are similar to this one.

GIT link: https://github.com/ashishdarekar/Kratos\_OpenFOAM\_adapter





# Dependencies / Requirements

- OpenFOAM-7 Fluid Solver (https://openfoam.org/download/7-ubuntu/)
- Kratos Structural Solver and "CoSimulation" Coupling tool ( https://github.com/KratosMultiphysics/Kratos) → Compile the Kratos using MPI (compilekratosmpi)
- CoSimIO (https://github.com/KratosMultiphysics/CoSimIO) → Build it, using the instructions given on the website(keeping MPI option ON) to generate the co\_sim\_io\_mpi.hpp file in <Directory\_Of\_CoSimIO>/co\_sim\_io
- Kratos-OpenFOAM adapter ( https://github.com/ashishdarekar/Kratos\_OpenFOAM\_adapter) → Configuration given in the next slide.
- Bash scripts for consistent & convenient use of Kratos ( https://github.com/philbucher/bash scripts)





# Bash File - for a reference

```
export PYTHONPATH=$PYTHONPATH:$HOME/Documents/MS/Kratos/Kratos/bin/Release
      export LD LIBRARY PATH=$LD LIBRARY PATH:$HOME/Documents/MS/Kratos/Kratos/bin/Release/libs
      #export KRATOS PATH ENV=~/Documents/MS/Kratos/Kratos
124 export KRATOS SOURCE=~/Documents/MS/Kratos/Kratos
125 alias startkratos="setupkratoseny /home/ashish/Documents/MS/Kratos/Kratos"
      . ~/Documents/MS/bash scripts/bash aliases common.sh
      . ~/Documents/MS/bash_scripts/bash_software.sh
      . ~/Documents/MS/bash scripts/bash kratos.sh
131 export PRECICE_ROOT=/home/ashish/Documents/MS/precice-2.2.0
     export Eigen3 ROOT=/home/ashish/eigen-3.3.9
      export PATH=$PATH:/path/to/extracted/location/version/bin
      export PKG CONFIG PATH=/home/ashish/Documents/MS/precice-2.2.0/build/lib/pkgconfig
      export LD LIBRARY PATH=$LD LIBRARY PATH:/home/ashish/Documents/MS/precice-2.2.0/build/lib
      export PATH="/home/ashish/Documents/MS/dealii-adapter/elasticity:${PATH}
      export PETSC DIR=/home/ashish/petscc
      export PETSC ARCH=arch-linux2-c-debug
      export LD LIBRARY PATH=$PETSC DIR/$PETSC ARCH/lib:$LD LIBRARY PATH
      source /opt/openfoam7/etc/bashrc
      #Some shortcuts
      export CDPATH=.:/home/ashish/Documents/MS/Kratos
      export PYTHONPATH=$PYTHONPATH:/home/ashish/Documents/MS/CoSimIO/bin
      export C INCLUDE PATH=$C INCLUDE PATH:/home/ashish/Documents/MS/CoSimIO/co sim io
      export CPLUS INCLUDE PATH=$CPLUS INCLUDE PATH:/home/ashish/Documents/MS/CoSimIO/co sim io
      export LD LIBRARY PATH=$LD LIBRARY PATH:/home/ashish/Documents/MS/CoSimIO/bin
      export CPLUS INCLUDE PATH=$CPLUS INCLUDE PATH:/home/ashish/Documents/MS/filesystem/include/
      export C INCLUDE PATH=$C INCLUDE PATH:/home/ashish/Documents/MS/Master thesis/Ashish/Thesis/OpenFoam Kratos adapter/Adapter
      export CPLUS INCLUDE PATH=$CPLUS INCLUDE PATH:/home/ashish/Documents/MS/Master thesis/Ashish/Thesis/OpenFoam Kratos adapter/Adapter
      export MPI_ROOT=/usr/lib/x86 64-linux-gnu/openmpi/include
      export OMPI MCA rmaps base oversubscribe=1 # Allow oversubscription for MPI (needed for OpenMPI >= 3.0)
```





# Configuration of the Adapter function object

- Clone the repository for Kratos-OpenFOAM Adapter https://github.com/ashishdarekar/Kratos\_OpenFOAM\_adapter
- 2) Go to the folder > KratosOpenfoamAdapterFunctionObject
- 3) Make linker related modifications in the file KratosOpenfoamAdapterFunctionObject/Make/options

```
-I$<Path_to_directory_CoSimIO>/co_sim_io \
-I$<Path_to_directory_CoSimIO> \
-L$<Path_to_directory_CoSimIO>/bin
```

4) MPI\_ROOT should direct to openmpi/include in your system

```
export MPI_ROOT=/usr/lib/x86_64-linux-gnu/openmpi/include
```

5) Give following commands to compile

```
wclean
wmake
```

6) It will generate "libKratosOpenfoamAdapterFunctionObjectFunctionObjects.so" shared library file in the same folder. One need to add this file in controlDict (OpenFOAM's config file) while using this function object.





# Make file for an adapter

```
KratosOpenfoamAdapterFunctionObject > Make > M options
      EXE INC = \
           -I$(LIB SRC)/finiteVolume/lnInclude \
           -I$(LIB SRC)/meshTools/lnInclude \
           -I$(LIB SRC)/transportModels/ \
           -I$(LIB SRC)/transportModels/incompressible/lnInclude \
           -I$(LIB SRC)/transportModels/compressible/lnInclude \
           -I$(LIB SRC)/transportModels/twoPhaseMixture/lnInclude \
           -I$(LIB SRC)/transportModels/interfaceProperties/lnInclude \
           -I$(LIB SRC)/transportModels/immiscibleIncompressibleTwoPhaseMixture/lnInclude \
           -I$(LIB SRC)/thermophysicalModels/basic/lnInclude \
                                                                                                      File structure and
           -I$(LIB SRC)/TurbulenceModels/turbulenceModels/lnInclude \
           -I$(LIB SRC)/TurbulenceModels/compressible/lnInclude \
                                                                                                      generated shared
           -I$(LIB SRC)/TurbulenceModels/incompressible/lnInclude \
                                                                                                      library file
           -I$(LIB SRC)/triSurface/lnInclude \
           -I$(MPI ROOT) \

    KratosOpenfoamAdapterFunctionObject

           -I$(MPI ROOT2) \
           -I$(HOME)/Documents/MS/CoSimIO/co sim io \
                                                                                > Introdude
           -I$(HOME)/Documents/MS/CoSimIO \
                                                                                > Make
           -L$(HOME)/Documents/MS/CoSimIO/bin
                                                                               KratosOpenfoamAdapterFunctionObject.C
      LIB LIBS = \
                                                                               C KratosOpenfoamAdapterFunctionObject.H
          -lfluidThermophysicalModels \
           -lincompressibleTransportModels \

≡ libKratosOpenfoamAdapterFunctionObjectFunctionObjects.so

           -lcompressibleTransportModels \
                                                                               > Tutorial_case_1_flap
           -lturbulenceModels \
           -lincompressibleTurbulenceModels \
                                                                               > Tutorial case 2 CAARC
           -lcompressibleTurbulenceModels \
                                                                               gitignore
           -limmiscibleIncompressibleTwoPhaseMixture \
                                                                               Abstract_of_Master_Thesis_ashish_darekar.pdf
           -lspecie \
           -lfileFormats \
                                                                                Master Thesis Presentation Ashish.pdf
           -lfiniteVolume \
                                                                              openfoam_wrapper.py
           -lmeshTools \
           -lco sim io mpi

 README.md
```



# Wind Generation as a B.C to OpenFOAM

```
∨ OPEN EDITORS
                                             Tutorial_case_2_CAARC > 0 > @ U > ...
                                                    rudilirite
      decomposeParDict Tutorial_case_2_C...
     (1) README.md
                                                        version
                                                                     2.0;
                                                        format
                                                                     ascii;
     {} ProjectParametersCoSim.json T... 2
                                                                     volVectorField:
   X 🕒 U Tutorial_case_2_CAARC/0
                                                        location
                                                                     "0";
                                                        object
      {} ProjectParametersCoSim.json Tutori...
      {} ProjectParametersCSD.json Tu... 9+
V OPENFOAM KRATOS ADAPTER
  > Tutorial_case_1_flap
                                                    dimensions
                                                                     [0 1 -1 0 0 0 0];

▼ Tutorial_case_2_CAARC

                                                    internalField
                                                                     uniform (11.11 0 0):
   V 0
                                                    //internalField uniform (0 0 0);
    ≣ k
                                                    boundaryField
    ≡ nuSqs
    ≡ nut
                                                        inlet
    ≡ nuTilda
    ≣ p

≡ pointDisplacement

                                                                             timeVaryingMappedFixedValue;
                                                            type
                                                                             timeVaryingMappedHDF5FixedValue
    C. U
                                                            setAverage
                                                                             Θ;
    ≡ windgen.h5
                                                            offset
                                                                             (0 0 0);
   > constant
                                                            perturb
                                                            hdf5FileName
                                                                             windgen.h5;
   > system
                                                            hdf5PointsDatasetName
   $ clean.sh
                                                            hdf5SampleTimesDatasetName
                                                            hdf5FieldValuesDatasetName
                                                                                             velocity;
  compute_base_reaction_process.py
  compute global force process.py
  compute_level_force_process.py
                                                        upperWall
  compute_total_force_process.py
                                                                             fixedValue:
                                                             type
  impose_custom_inlet_velocity_process.py
                                                            value
                                                                             uniform (11.11 0 0);
    inlet velocity 2D function implementa
```

WindGen inlet linked to Boundary condition in OF





# Configuration file for CoSimulation

```
EXPLORER
                                                 {} ProjectParametersCoSim.json 2 X
                                                 Tutorial case 2 CAARC > {} ProjectParametersCoSim.json > {} solver settings > [ ] coupling sequence > {} 0

∨ OPEN EDITORS

                                                                         "solver"
                                                                                             "load interface 3D structure",
     V OPENFOAM_KRATOS_ADAPTER ☐ ☐ ♡ ☐
                                                                         "scaling factor" : "(sin(((t)/0.001)/10000*pi/2))**2 ",
       > KratosOpenfoamAdapterFunctionObject
                                                                         "echo level"
                                                                         "interval"
                                                                                           : [0.0, 10.0]
       > Tutorial case 1 flap
         ≣ k
"Openfoam Kratos Wrapper":
         ≡ nuSas
         ≣ nut
                                                                         "solver wrapper settings" : {
         ≡ nuTilda
                                                                             "import meshes"
                                                                             "export data"
                                                                             "import data"
                                                                                                 : ["load interface 3D structure"]

    ■ pointDisplacement

                                                                                          : "kratos_co_sim_io",
         ≡ windgen.h5
                                                                             "echo level" : 1,
         > constant
                                                                             "connect to" : "Openfoam Adapter",
                                                                             "communication format" : "file"
        > system
        $ clean.sh
                                                                         "data" : {
        compute_base_reaction_process.py
                                                                             "disp interface 3D structure" : {
                                                                                 "model part name" : "interface 3D structure",
        compute_global_force_process.py
                                                                                 "variable_name" : "MESH_DISPLACEMENT",
        compute_level_force_process.py
                                                                                  "dimension"
        compute total force process.py
                                                                             "load interface 3D structure" : {
        impose_custom_inlet_velocity_process.py
                                                                                 "model_part_name"
                                                                                                     : "interface_3D_structure",
        inlet_velocity_2D_function_implementa...
                                                                                 "dimension" : 3,
                                                                                 "variable name" : "REACTION"
        line output process.py
        MainKratosCoSim.py
        ≡ mesh CSD.mdpa
        multiple_points_output_process.py
        output_sum_forces_process.py
        plot displacements.pv
                                                                         "solver_wrapper_settings" : {
                                                                             "input file" : "ProjectParametersCSD"
        point_output_process.py
        {} ProjectParametersCoSim.json
                                                                         "data" : {
        {} ProjectParametersCSD.json
                                                                             "disp_interface_3D_structure" : {
                                                                                  "model_part_name" : "Structure.PointLoad3D hull points",
        {} StructuralMaterials.ison
                                                                                 "variable name" : "DISPLACEMENT",
       gitignore
                                                                                 "dimension"
       Abstract_of_Master_Thesis_ashish_darek...
                                                                             "load interface 3D structure" : {
       Master_Thesis_Presentation_Ashish.pdf
                                                                                  "model part name" : "Structure.PointLoad3D hull points",
       openfoam_wrapper.py
                                                                                 "variable name" : "POINT LOAD",
                                                                                 "dimension"

 README.md
```

This is provided in the file called

ProjectParametersCoSim .json

### **Detailed File:**

https://github.com/ashishd arekar/Kratos\_OpenFOAM\_adapter/blob/main/Tutoria \_\_case\_2\_CAARC/Project ParametersCoSim.json





# Configuration file for OpenFOAM

```
∨ OPENFOAM_KRATOS_ADAPTER

                                                        timePrecision
        > .vscode
       > KratosOpenfoamAdapterFunctionObject
                                                        functions
       > Tutorial case 1 flap

▼ Tutorial_case_2_CAARC

                                                            // Function object "KratosOpenfoamAdapterFunctionObject" to couple OpenFOAM with CoSimulation (kratos)
                                                            KratosOpenfoamAdapterFunctionObject1
        > 0
        > constant
                                                                type KratosOpenfoamAdapterFunctionObject;
                                                                libs ("../KratosOpenfoamAdapterFunctionObject/libKratosOpenfoamAdapterFunctionObjectFunctionObjects.so")
出

    ■ blockMeshDict

                                                                interfaces
         @ controlDict

    ■ decomposeParDict

                                                                    Interface1

≡ fvSchemes

                                                                                                 interface 3D structure;
          (tubesfs);
                                                                        patches
                                                                        importDataIdentifier
                                                                                                 (disp interface 3D structure);

■ probes

                                                                        exportDataIdentifier
                                                                                                 (load interface 3D structure);

≡ sample

≡ surfaceFeatureExtractDict

                                                                parameters
        $ clean.sh
                                                                    rho rho [1 -3 0 0 0 0 0] 1.2;
        compute_base_reaction_process.py
                                                                    nu nu [0 2 -1 0 0 0 0 ] 1.508e-05;
        compute_global_force_process.py
        compute_level_force_process.py
        compute total force process.py
                                                                debugLevel debug;
        impose_custom_inlet_velocity_process.py
```

# This is provided in the file called **system/controlDict** Detailed File:

https://github.com/ashishdarekar/Kratos\_OpenFOAM\_adapter/blob/main/Tutorial\_case\_2\_CAARC/system/controlDict





- Go to the folder > Tutorial\_case\_2\_CAARC
- 2) Simultaniuosly run 2 terminals, one for OpenFOAM and another for Kratos.
- 3) Run following commands of OpenFOAM on 1st terminal

```
blockMesh
decomposePar
mv 0/ 0_org/
mkdir 0
mpirun -np <number_of_processes> snappyHexMesh -overwrite -parallel
reconstructParMesh -constant
rm -r 0/ processor0/ processor1/ processor2/ ..... processor<number_of_processes-1>
mv 0_org/ 0/
decomposePar
mpirun -np <number_of_processes> pimpleFoam -parallel
```

Note: Number of processors need to mention in the file decomposeParDict





->>>Setting Number of MPI cores, and (x,y,z) configuration

```
EXPLORER
                                        decomposeParDict X
V OPEN EDITORS
                                        Tutorial_case_2_CAARC > system > @ decomposeParDict
  X G decomposeParDict Tutorial case 2 C...
V OPENFOAM_KRATOS_ADAPTER ☐ ☐ ☐ ☐
                                                // / F ield | OpenFOAM: The Open Source CFD Toolbox
 > .vscode
 > KratosOpenfoamAdapterFunctionObject
 > Tutorial_case_1_flap
                                              FoamFile

→ Tutorial_case_2_CAARC

  > 0
                                                  version
                                                             2.0;
                                                  format
                                                             ascii;
                                                             dictionary;
  > constant
                                                  location "system";
  > processor0
                                                              decomposeParDict;
  > processor1

∨ system

≡ blockMeshDict

                                              numberOfSubdomains 2;
   method
                                                             simple;
   G decomposeParDict

≡ fvSchemes

                                              simpleCoeffs
   ≡ fvSolution
                                                                 (211);
   ≡ probes
                                                  delta
                                                                 0.001;

≡ sample

≡ surfaceFeatureExtractDict

  compute_base_reaction_process.py
  compute_global_force_process.py
  compute_level_force_process.py
  compute_total_force_process.py
```





-->>> Example Output of this terminal

```
No MRF models present
No finite volume options present
Constructing face velocity Uf
Courant Number mean: 0.000440997277485 max: 0.0148631527897
Starting time loop
[0] [AdapterInfo] CoSimulation Adapter's function object : read()
[0] [AdapterInfo] Dimension of a problem is: 3
[0] [AdapterInfo] Determining the solver type
[1] [AdapterInfo] CoSimulation Adapter's function object : read()
[1] [AdapterInfo] Dimension of a problem is: 3
[1] [AdapterInfo] Determining the solver type
[1] [AdapterInfo] Solver Type : InCompressible
[1] [AdapterInfo] Coupling Interfaces Reading: Start
[1] [AdapterInfo] Name of the coupling interface is = interface 3D structure
[1] [AdapterInfo] Coupling Interfaces Reading: Done
[1] [AdapterInfo] Number of coupling interfaces found: 1
[0] [AdapterInfo] Solver Type : InCompressible
[0] [AdapterInfo] Coupling Interfaces Reading: Start
[0] [AdapterInfo] Name of the coupling interface is = interface 3D structure
[0] [AdapterInfo] Coupling Interfaces Reading: Done
[0] [AdapterInfo] Number of coupling interfaces found: 1
CoSimIO: CoSimIO from "Openfoam Adapter" to "Openfoam Kratos Wrapper" uses communication format: file
```





4) Run following commands of Kratos on 2nd terminal

```
startkratos
runkratosmpi MainKratosCoSim.py <number_of_processes>
```

### -->>> Example Output of this terminal

```
(base) ashish:~/.../OpenFoam_Kratos_adapter/Tutorial_case_2_CAARC$ runkratosmpi MainKratosCoSim.py 2
Kratos Path: /home/ashish/Documents/MS/Kratos/Kratos
Compiled Configuration Information:
Branch:
                  cosim/openfoam-wrapper
Version:
==== PARALLEL EXECUTION =====
with 2 processes (and setting OMP NUM THREADS=1)
            Multi-Physics 9.0."3"-ab55314978-FullDebug
Compiled with threading and MPI support.
Maximum number of threads: 1.
MPI world size:
Process Id: 14752
Importing KratosCoSimulationApplication
Initializing KratosCoSimulationApplication...
COSIMTOOLS: Created ModelPart "interface 3D structure" for solver "Openfoam Kratos_Wrapper"

COSIMTOOLS: Allocating historical variable "MESH_DISPLACEMENT" in ModelPart "interface_3D_structure" for solver "Openfoam_Kratos_Wrapper"
CoSimTools: Allocating historical variable "REACTION" in ModelPart "interface_3D_structure" for solver "Openfoam_Kratos_Wrapper'
Importing
             KratosStructuralMechanicsApplication
Initializing KratosStructuralMechanicsApplication...
[WARNING] Parallel Type: "OpenMP" is specified as "parallel_type", but Kratos is running distributed!
::[MechanicalSolver]:: : Construction finished
::[ImplicitMechanicalSolver]:: : Construction finished
::[MechanicalSolver]:: : Variables ADDED
::[ImplicitMechanicalSolver]:: : Variables ADDED
```





5) To see the results in ParaView:

```
a. reconstructPar
paraFoam -case .&
```

- b. Load the KRATOS results from the folder > vtk output structure
- 6) To clean all the generated files: Run the script *clean.sh*

```
./clean.sh
```

#### **Details on GIT:**

https://github.com/ashishdarekar/Kratos OpenFOAM adapter





### **Everything is running:** Example Output

```
Courant Number mean: 0.000494505659341 max: 0.0432009327319
                                                                                                    [WARNING] Triangle3D3: The [3](0,0,0) is in a distance: 7.5
Time = 0.005
                                                                                                    [WARNING] Triangle3D3: The [3](0,0,0) is in a distance: 7.5
                                                                                                    [WARNING] BruteForcePointLocator: No Element found for Point: Point (0, 0, 0)
smoothSolver: Solving for cellDisplacementx, Initial residual = 0.329470465843, Final residual =
                                                                                                   [WARNING] PointOutputProcess: No "element" was found for input Point (0, 0, 0), no outp
9.99670881092e-09, No Iterations 641
smoothSolver: Solving for cellDisplacementy, Initial residual = 0.332528719429, Final residual =
                                                                                                   KratosMappingDataTransferOperator: Creating Mapper:
9.8937227639e-09, No Iterations 360
                                                                                                       Origin: ModelPart "interface_3D_structure" of solver "Openfoam_Kratos_Wrapper"
smoothSolver: Solving for cellDisplacementz, Initial residual = 0.206713<u>262655, Final residual =</u>
                                                                                                       Destination: ModelPart "Structure.PointLoad3D hull points" of solver "structure"
1.0306885292e-08, No Iterations 1000
                                                                                                                KratosMetisApplication
                                                                                                   Importing
GAMG: Solving for pcorr, Initial residual = 1, Final residual = 8.71091533989e-06, No Iterations
                                                                                                       KRATOS
GAMG: Solving for pcorr, Initial residual = 0.00741802481745, Final <math>residual = 8.05134001017e-06,
No Iterations 7
time step continuity errors : sum local = 1.0483467722e-17, global = -4.82361517556e-19, cumulativ
smoothSolver: Solving for Ux, Initial residual = 0.0010939825854, Final residual = 5.74039622422e Importing
smoothSolver: Solving for Uy, Initial residual = 0.000878467117207, Final residual = 7.7154203681
7e-09, No Iterations 1
smoothSolver: Solving for Uz, Initial residual = 0.000448565189422, Final residual = 1.8162405982
8e-09, No Iterations 1
GAMG: Solving for p, Initial residual = 0.160369097315, Final residual = 0.0123742024592, No Iter Initializing KratosTrilinosApplication...
                                                                                                   ScalingOperation: Scaling-Factor 2.4674010799787785e-08
GAMG: Solving for p, Initial residual = 0.0161304183063, Final residual = 0.00141930700772, No It KratosMappingDataTransferOperator: Creating Mapper:
                                                                                                       Origin: ModelPart "Structure.PointLoad3D_hull_points" of solver "structure"
erations 2
time step continuity errors : sum local = 1.12399566177e-10, global = 1.42751521835e-12, cumulativ
                                                                                                       Destination: ModelPart "interface 3D structure" of solver "Openfoam Kratos Wrapper
GAMG: Solving for p, Initial residual = 0.00271550271668, Final residual = 0.000256480491772, No
                                                                                                   time=0.002 | step=2:
GAMG: Solving for p, Initial residual = 0.000450613887124, Final residual = 8.02464500532e-07, No ::[KSM Simulation]:: : STEP: 2
Iterations 10
                                                                                                   ::[KSM Simulation]:: : TIME: 0.002
time step continuity errors : sum local = 6.24429089988e-14, global = -6.01485857951e-15, cumulati ScalingOperation: Scaling-Factor 9.86960407639239e-08
ve = -2.16174003774e-09
smoothSolver: Solving for k, Initial residual = 0.0725375677349, Final residual = 5.4233<u>189913e-1 time=0.003 | step=3:</u>
1. No Iterations 2
                                                                                                   ::[KSM Simulation]:: : STEP: 3
                                                                                                   ::[KSM Simulation]:: : TIME: 0.003
ExecutionTime = 877.52 s ClockTime = 6118 s
                                                                                                   ScalingOperation: Scaling-Factor 2.2206608258672686e-07
[0] [AdapterInfo] 0.005 : CoSimulation Adapter's function object : execution()
[0] [AdapterInfo] Force calculation started for coupling interface: interface 3D structure
                                                                                                   time=0.004 | step=4:
[1] [AdapterInfo] 0.005 : CoSimulation Adapter's function object : execution()
                                                                                                   ::[KSM Simulation]:: : STEP: 4
[1] [AdapterInfo] Force calculation started for coupling interface: interface_3D_structure
                                                                                                   ::[KSM Simulation]:: : TIME: 0.004
[0] [AdapterInfo] Conversion of Elemental Force to Nodal Force: interface 3D structure
                                                                                                   ScalingOperation: Scaling-Factor 3.947841240920618e-07
[1] [AdapterInfo] Conversion of Elemental Force to Nodal Force: interface_3D_structure
[0] [AdapterInfo] Data has been exported from OpenFOAM to CoSimulation (for coupling interface nam time=0.005 | step=5:
 = interface_3D_structure)
                                                                                                   ::[KSM Simulation]:: : STEP: 5
[1] [AdapterInfo] Data has been exported from OpenFOAM to CoSimulation (for coupling interface nam ::[KSM Simulation]:: : TIME: 0.005
 = interface_3D_structure)
                                                                                                   ScalingOperation: Scaling-Factor 6.168501482333415e-07
```