Lehrstuhl für Statik
Ingenieurfakultät Bau Geo Umwelt
Technische Universität München

Tutorial:

CAARC building-FSI simulation OpenFOAM-CoSimulation-Kratos

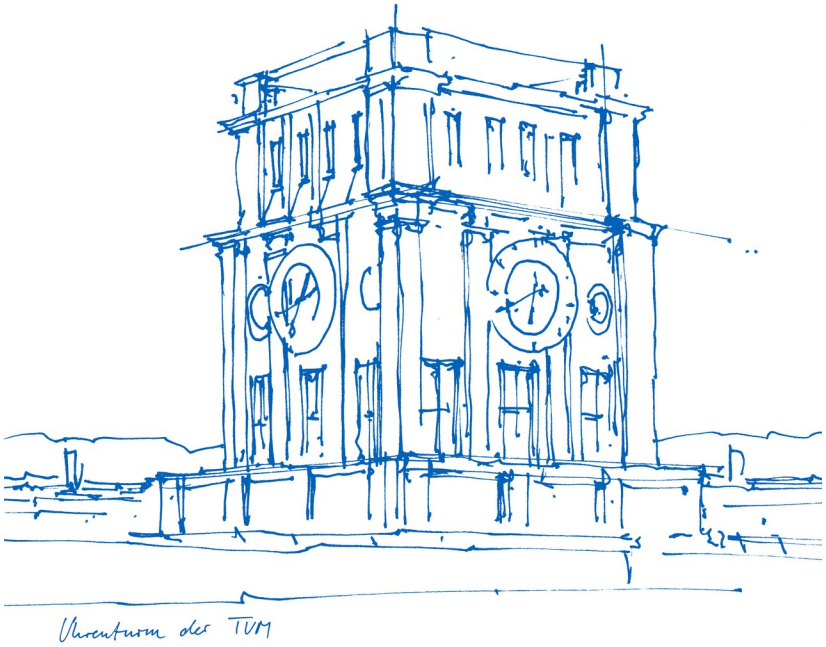
Ashish Shankar, Darekar

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

```
#Kratos
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
export KRATOS_SOURCE=~/Documents/MS/Kratos/Kratos
alias startkratos="setupkratosenv /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

#PreCICE
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}"

#PETSc
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

#OpenFOAM
source /opt/openfoam7/etc/bashrc

#Some_shortcuts
export CDPATH=.:/home/ashish/Documents/MS/Kratos

#CoSimIO
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

#Filesystem
export CPLUS_INCLUDE_PATH=$CPLUS_INCLUDE_PATH:/home/ashish/Documents/MS/filesystem/include/

#CoSimulationAdapter
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

#MPI
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

1) 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
```

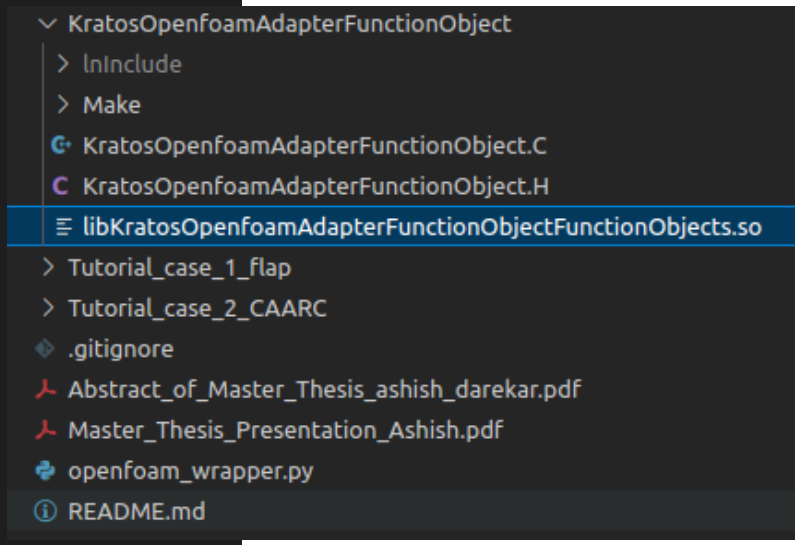
6) It will generate "*libKratosOpenfoamAdapterFunctionObjectFunctionObjects.so*" shared library file in the same folder. One need to add this file in controlDict (OpenFOAM‘s config file) while using this function object.

Make file for an adapter

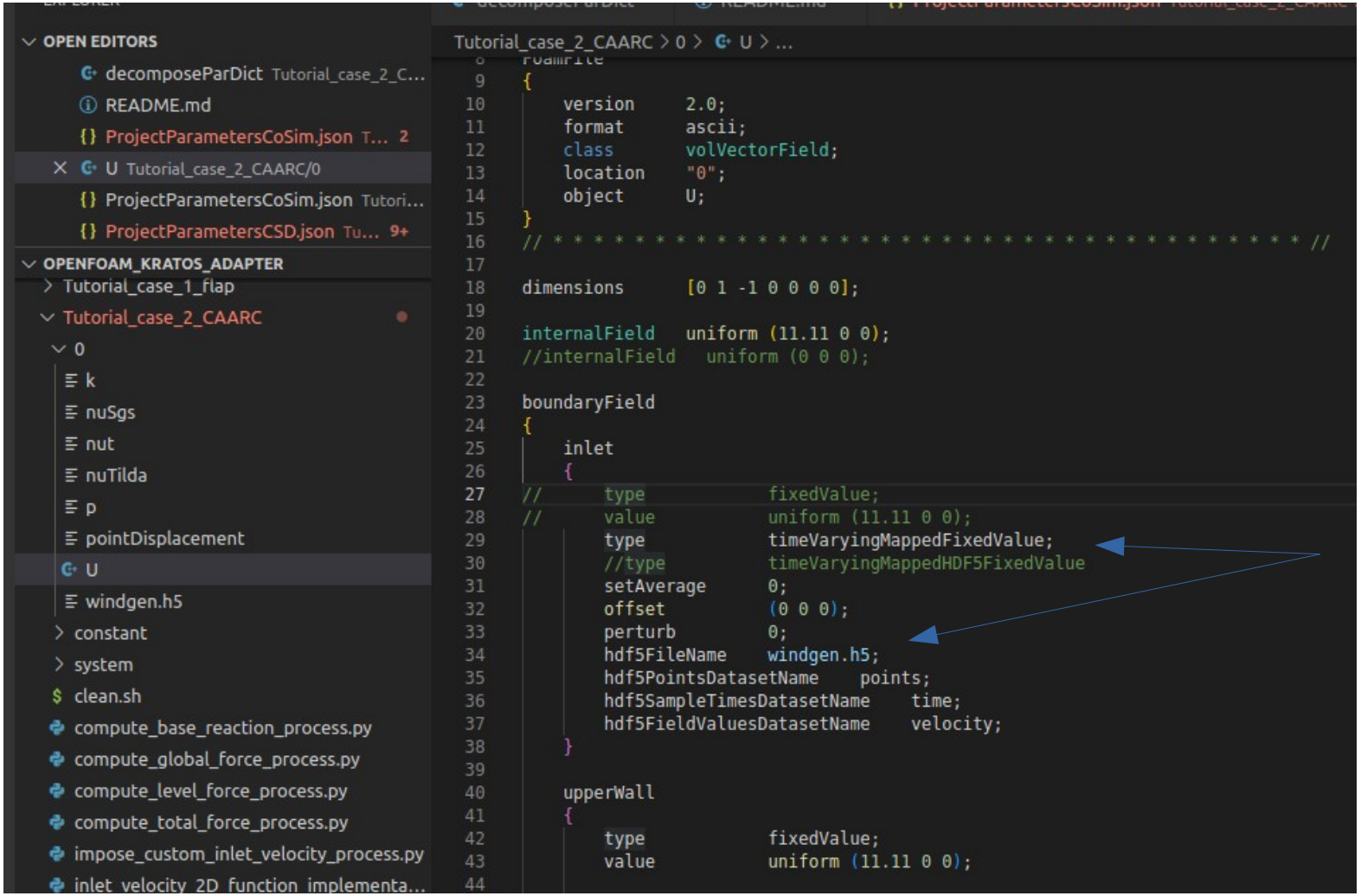
KratosOpenfoamAdapterFunctionObject > Make > M options

```
1   EXE_INC = \
2       -I$(LIB_SRC)/finiteVolume/lnInclude \
3       -I$(LIB_SRC)/meshTools/lnInclude \
4       -I$(LIB_SRC)/transportModels/ \
5       -I$(LIB_SRC)/transportModels/incompressible/lnInclude \
6       -I$(LIB_SRC)/transportModels/compressible/lnInclude \
7       -I$(LIB_SRC)/transportModels/twoPhaseMixture/lnInclude \
8       -I$(LIB_SRC)/transportModels/interfaceProperties/lnInclude \
9       -I$(LIB_SRC)/transportModels/immiscibleIncompressibleTwoPhaseMixture/lnInclude \
10      -I$(LIB_SRC)/thermophysicalModels/basic/lnInclude \
11      -I$(LIB_SRC)/TurbulenceModels/turbulenceModels/lnInclude \
12      -I$(LIB_SRC)/TurbulenceModels/compressible/lnInclude \
13      -I$(LIB_SRC)/TurbulenceModels/incompressible/lnInclude \
14      -I$(LIB_SRC)/triSurface/lnInclude \
15      -I$(MPI_ROOT) \
16      -I$(MPI_ROOT2) \
17      -I$(HOME)/Documents/MS/CoSimIO/co_sim_io \
18      -I$(HOME)/Documents/MS/CoSimIO \
19      -L$(HOME)/Documents/MS/CoSimIO/bin
20
21  LIB_LIBS = \
22      -lfluidThermophysicalModels \
23      -lincompressibleTransportModels \
24      -lcompressibleTransportModels \
25      -lturbulenceModels \
26      -lincompressibleTurbulenceModels \
27      -lcompressibleTurbulenceModels \
28      -limmiscibleIncompressibleTwoPhaseMixture \
29      -lspecie \
30      -lfileFormats \
31      -lfiniteVolume \
32      -lmeshTools \
33      -lco_sim_io_mpi
34
```

File structure and generated shared library file

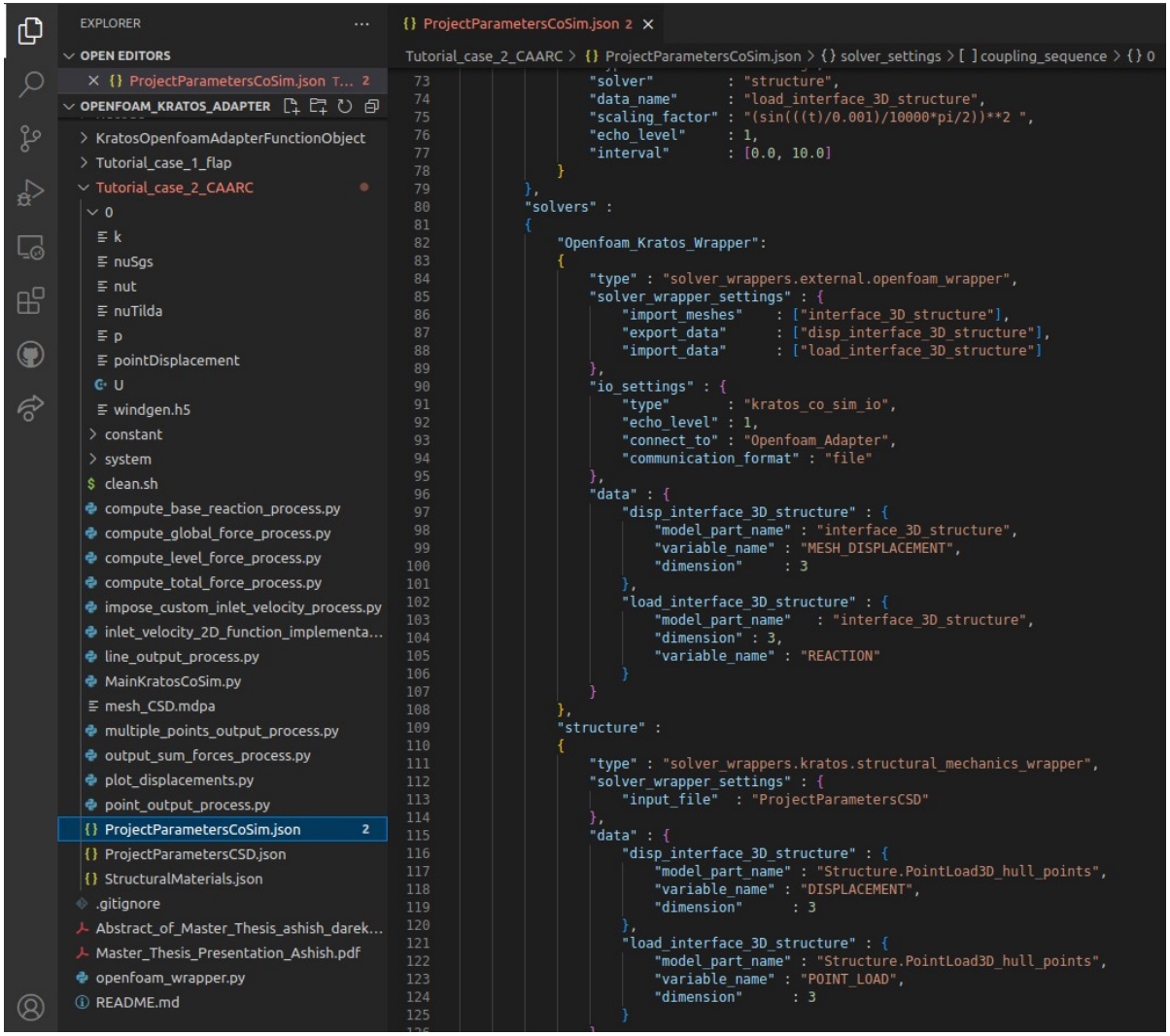


Wind Generation as a B.C to OpenFOAM



WindGen inlet linked to Boundary condition in OF

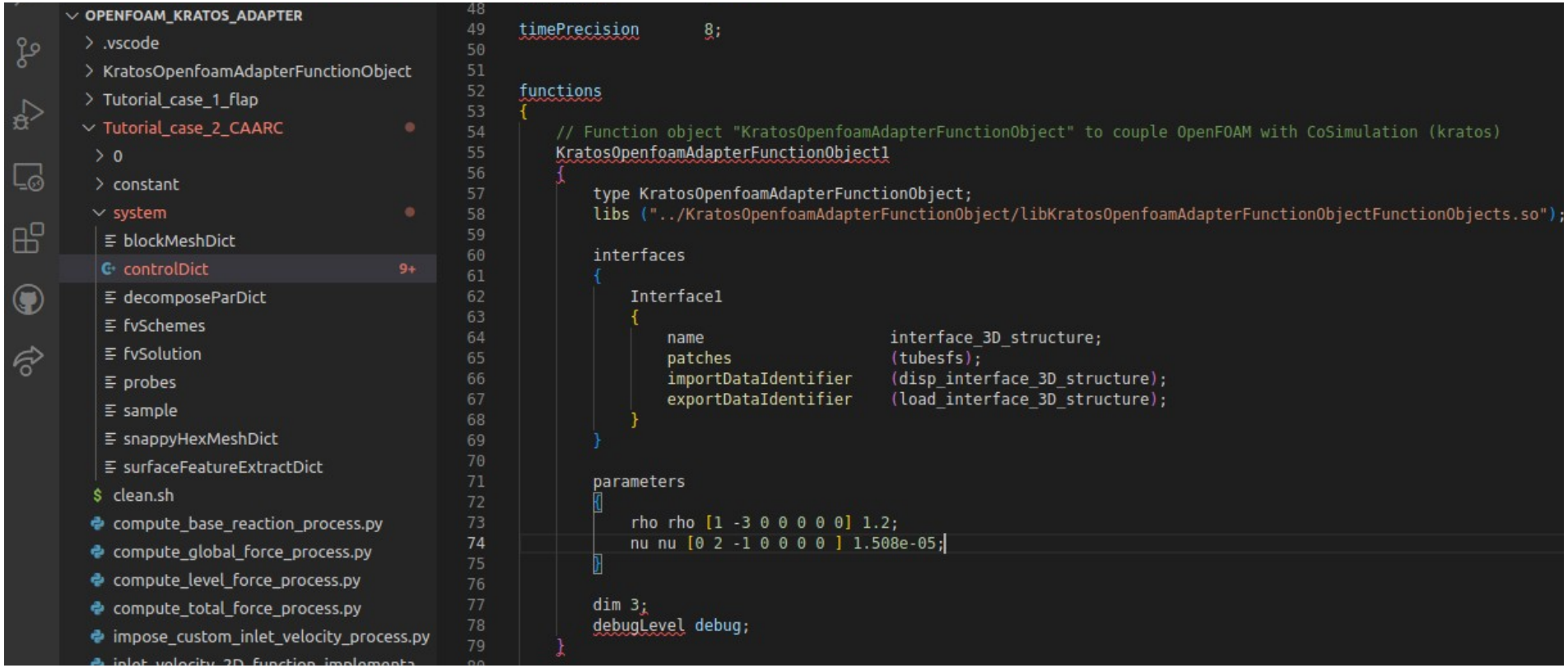
Configuration file for CoSimulation



This is provided in the file called ***ProjectParametersCoSim.json***

Detailed File:
https://github.com/ashishdarekar/Kratos_OpenFOAM_adapter/blob/main/Tutorial_case_2_CAARC/ProjectParametersCoSim.json

Configuration file for OpenFOAM



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

To run the simulation – With MPI only

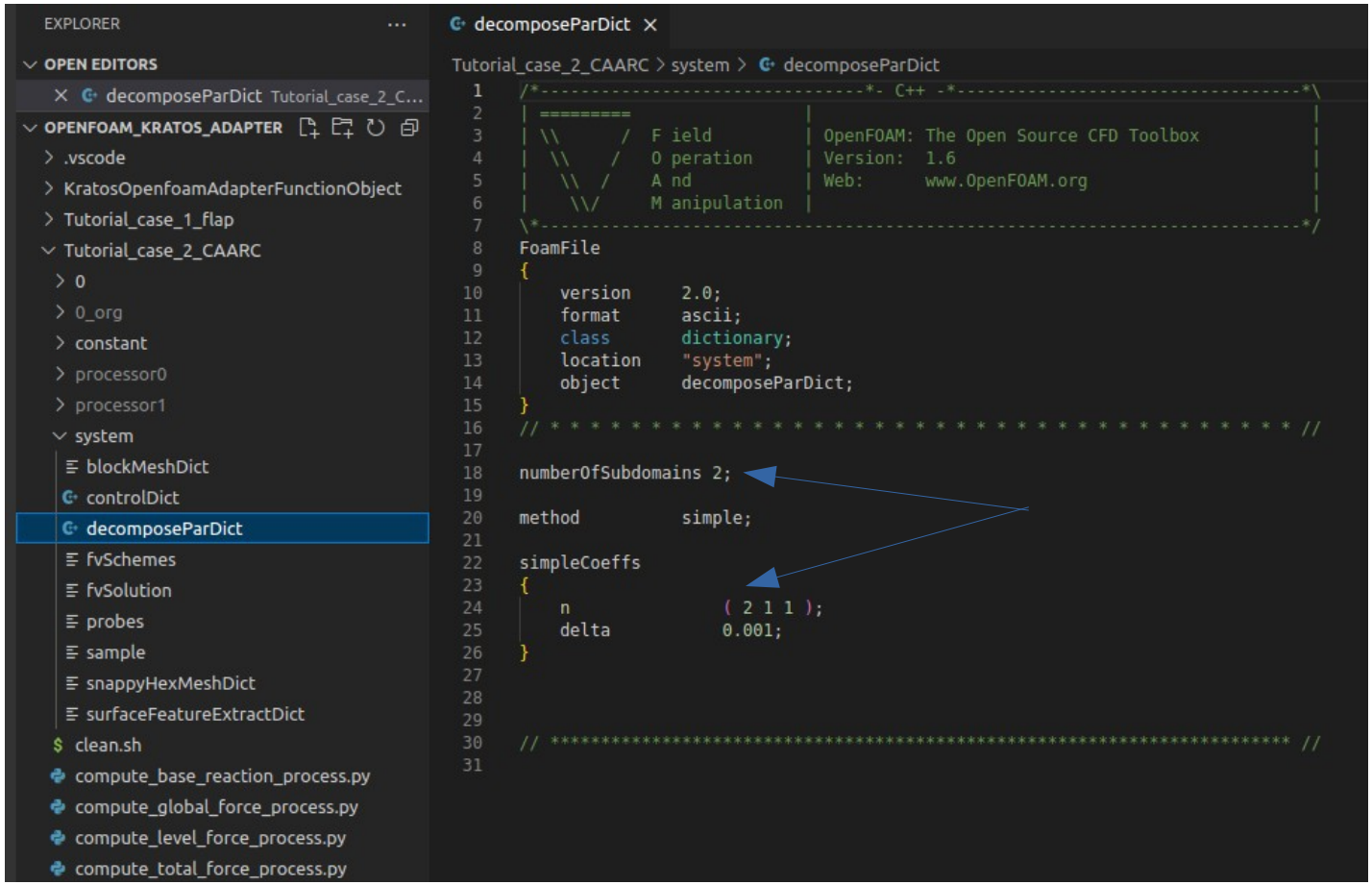
1) Go to the folder > Tutorial_case_2_CAARC
2) Simultaniuosly run 2 terminals, one for OpenFOAM and another for Kratos.
3) Run following commands of OpenFOAM on 1st terminal

```
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*

To run the simulation – With MPI only

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



To run the simulation – With MPI only

-->>>> 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
```

To run the simulation – With MPI only

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$ startkratos
 Info: Started Kratos /home/ashish/Documents/MS/Kratos/Kratos
(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:         mpi
Mode:            FullDebug
===== 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:       2.
Process Id: 14752
Importing    KratosCoSimulationApplication
    KRATOS  / ___|___/ ___|(_)_ __ ___  _   _| | __ _| |_(_) ___  _ __
           | |   / _ \___ \| | '_ ` _ \| | | | |/ _` | __| |/ _ \| '_ \
           | |__| (_) |__) | | | | | | | |_| | | (_| | |_| | (_) | | | |
            \____\___/____/|_|_| |_| |_|_|\__,_|_|\__,_|\__|_|\___/|_| |_|
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
    KRATOS    ___|  |                   |                   |
             \___ \  __|  __| |   |  __| __| |   |  __| _` | |
                   | |   |    |   | (    |   |   | |   (   | |
             _____/ \__|_|  \__,_|\___|\__|\__,_|_|  \__,_|_| MECHANICS
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
```

To run the simulation – With MPI only

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

To run the simulation – With MPI only

Everything is running: Example Output

```
Courant Number mean: 0.000494505659341 max: 0.0432009327319
Time = 0.005

smoothSolver:  Solving for cellDisplacementx, Initial residual = 0.329470465843, Final residual =
9.99670881092e-09, No Iterations 641
smoothSolver:  Solving for cellDisplacementy, Initial residual = 0.332528719429, Final residual =
9.8937227639e-09, No Iterations 360
smoothSolver:  Solving for cellDisplacementz, Initial residual = 0.206713262655, Final residual =
1.0306885292e-08, No Iterations 1000
GAMG:  Solving for pcorr, Initial residual = 1, Final residual = 8.71091533989e-06, No Iterations
23
GAMG:  Solving for pcorr, Initial residual = 0.00741802481745, Final residual = 8.05134001017e-06,
 No Iterations 7
time step continuity errors : sum local = 1.0483467722e-17, global = -4.82361517556e-19, cumulativ
e = -2.1631615381e-09
smoothSolver:  Solving for Ux, Initial residual = 0.0010939825854, Final residual = 5.74039622422e
-09, No Iterations 1
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
ations 1
GAMG:  Solving for p, Initial residual = 0.0161304183063, Final residual = 0.00141930700772, No It
erations 2
time step continuity errors : sum local = 1.12399566177e-10, global = 1.42751521835e-12, cumulativ
e = -2.16173402288e-09
GAMG:  Solving for p, Initial residual = 0.00271550271668, Final residual = 0.000256480491772, No
Iterations 2
GAMG:  Solving for p, Initial residual = 0.000450613887124, Final residual = 8.02464500532e-07, No
 Iterations 10
time step continuity errors : sum local = 6.24429089988e-14, global = -6.01485857951e-15, cumulati
ve = -2.16174003774e-09
smoothSolver:  Solving for k, Initial residual = 0.0725375677349, Final residual = 5.4233189913e-1
1, No Iterations 2
ExecutionTime = 877.52 s  ClockTime = 6118 s

[0] [AdapterInfo] 0.005 : CoSimulation Adapter's function object : execution()
[0] [AdapterInfo] Force calculation started for coupling interface : interface_3D_structure
[1] [AdapterInfo] 0.005 : CoSimulation Adapter's function object : execution()
[1] [AdapterInfo] Force calculation started for coupling interface : interface_3D_structure
[0] [AdapterInfo] Conversion of Elemental Force to Nodal Force : interface_3D_structure
[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
e = interface_3D_structure)
[1] [AdapterInfo] Data has been exported from OpenFOAM to CoSimulation (for coupling interface nam
e = interface_3D_structure)
```

```
[WARNING] Triangle3D3: The [3](0,0,0) is in a distance: 7.5
[WARNING] Triangle3D3: The [3](0,0,0) is in a distance: 7.5
[WARNING] BruteForcePointLocator: No Element found for Point: Point (0, 0, 0)
[WARNING] PointOutputProcess: No "element" was found for input Point (0, 0, 0), no outp
ut is written!
KratosMappingDataTransferOperator: Creating Mapper:
    Origin: ModelPart "interface_3D_structure" of solver "Openfoam_Kratos_Wrapper"
    Destination: ModelPart "Structure.PointLoad3D_hull_points" of solver "structure"
Importing    KratosMetisApplication
    KRATOS   __  __      _   _
            |  \/  | ___| |_(_)___
            | |\/| |/ _ \ __| / __|
            | |  | |  __/ |_| \__ \
            |_|  |_|\___|\__|_|___/
Initializing KratosMetisApplication...
Importing    KratosTrilinosApplication
    KRATOS   _____      _ _ _
            |_   _| __(_) (_)_ __   ___  ___
              | || '__| | | | '_ \ / _ \/ __|
              | || |  | | | | | | | (_) \__ \
              |_||_|  |_|_|_|_| |_|\___/|___/
Initializing KratosTrilinosApplication...
ScalingOperation: Scaling-Factor 2.4674010799787785e-08
KratosMappingDataTransferOperator: Creating Mapper:
    Origin: ModelPart "Structure.PointLoad3D_hull_points" of solver "structure"
    Destination: ModelPart "interface_3D_structure" of solver "Openfoam_Kratos_Wrapper"


time=0.002 | step=2:
::[KSM Simulation]:: : STEP:  2
::[KSM Simulation]:: : TIME:  0.002
ScalingOperation: Scaling-Factor 9.86960407639239e-08

time=0.003 | step=3:
::[KSM Simulation]:: : STEP:  3
::[KSM Simulation]:: : TIME:  0.003
ScalingOperation: Scaling-Factor 2.2206608258672686e-07

time=0.004 | step=4:
::[KSM Simulation]:: : STEP:  4
::[KSM Simulation]:: : TIME:  0.004
ScalingOperation: Scaling-Factor 3.947841240920618e-07

time=0.005 | step=5:
::[KSM Simulation]:: : STEP:  5
::[KSM Simulation]:: : TIME:  0.005
ScalingOperation: Scaling-Factor 6.168501482333415e-07
```