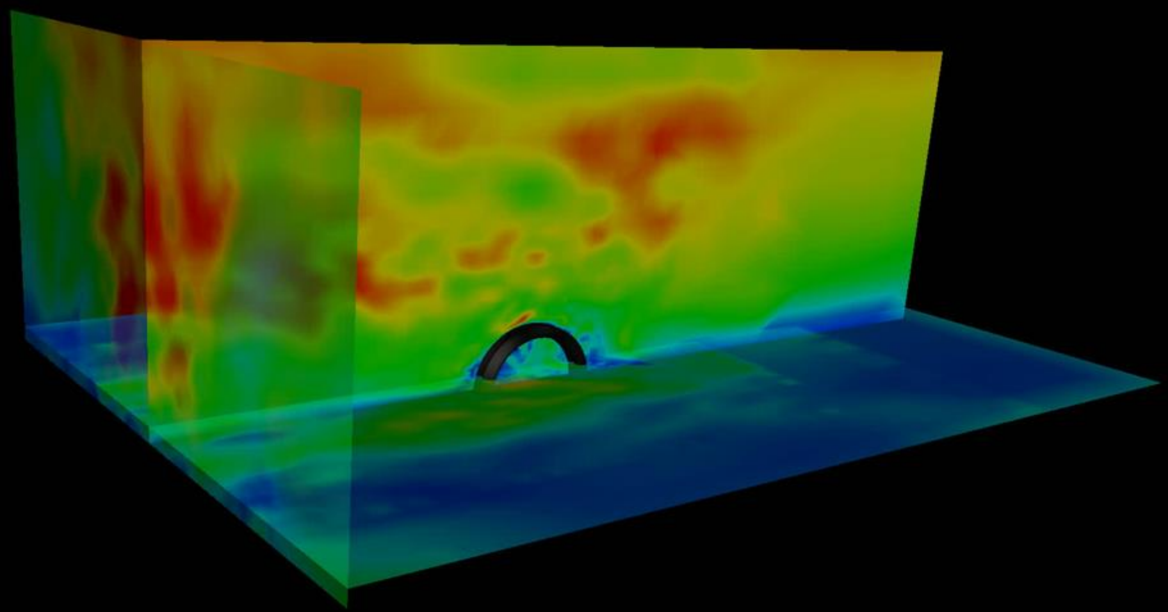
Chair of Structural Analysis
TUM Department of Civil, Geo and Environmental Engineering
Technical University of Munich



STRUCTURAL WIND ENGINEERING

Roland Wüchner, Chair of Structural Analysis, TUM
Máté Péntek, Chair of Structural Analysis, TUM

Presentation material from internal and external sources have been used either directly, modified or adapted to fit the purpose. Effort is continuously being made to accurately reference these. Nonetheless, check referencing in both the script as well as slides for completeness. In case of inconsistencies or mistakes please contact us!

Kratos 2D FSI Tutorial – Building

In this tutorial a fully coupled Fluid-Structure-Interaction (FSI) simulation is introduced. The key points of the simulation are pointed out. The simulation will be executed and the results will be reviewed

Covered topics:

- Problem description of FSI
- Predefined FSI example
- Postprocessing of results
- Some theoretical aspects of a FSI-simulation

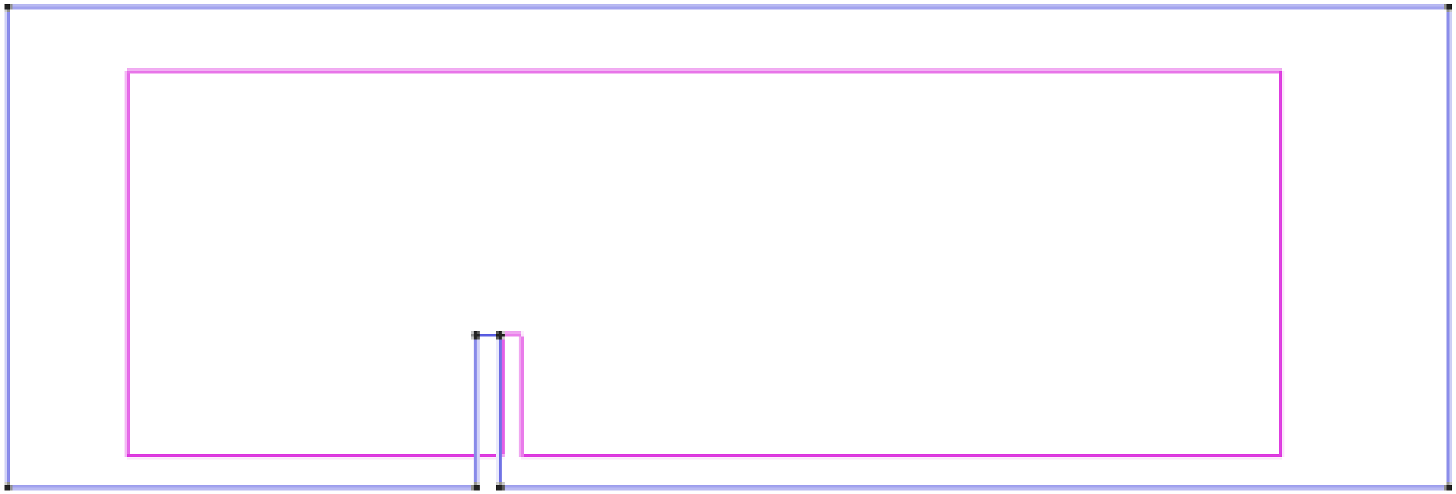
Disclaimer: This example serves the sole educational purpose of demonstrating how to setup a basic 2D FSI problem, run the simulation and do some postprocessing. For any real case in wind engineering a 3D setup should be adopted accompanied with detailed mesh and time step study.

Technical note: Tested on 04.12.2019, works with GiD 14.1.7d and the pre-release of the Kratos problemtype (7.1) on Windows 10 and Ubuntu 18 64 bit.

Problem description

- The problem investigated in this tutorial is a 2D building in a 2D fluid domain. The building is excited / deformed by the flow surrounding it.

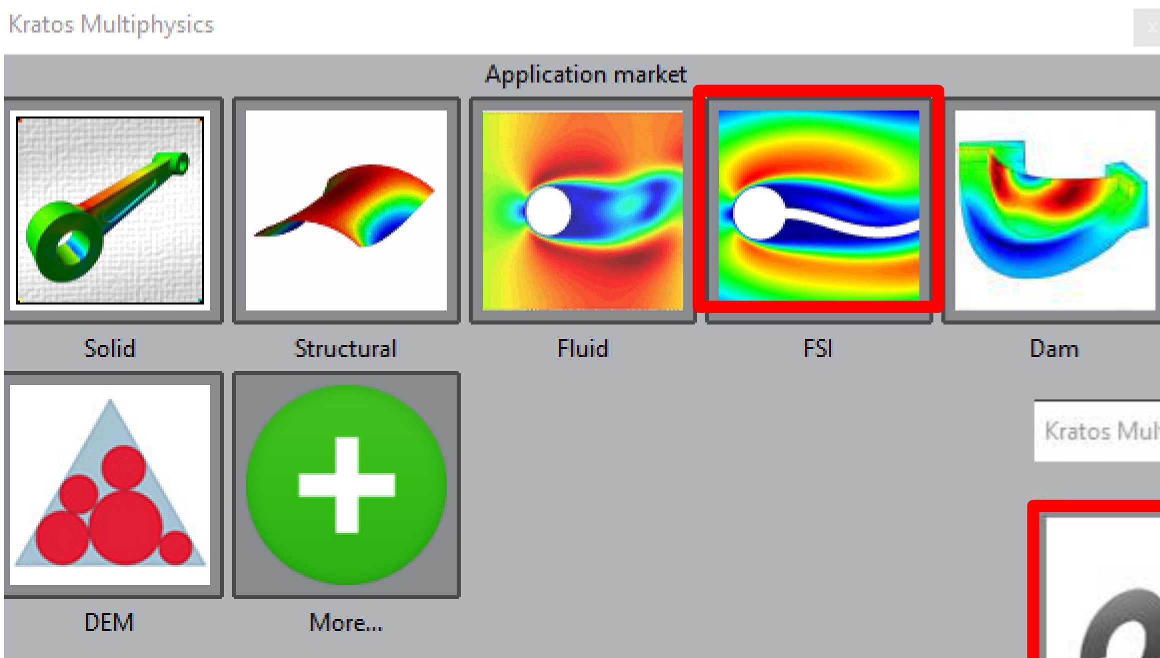
- The geometry is same as was used in the second tutorial related to computation methods: → *Tutorial4_2D_CFD.gid*

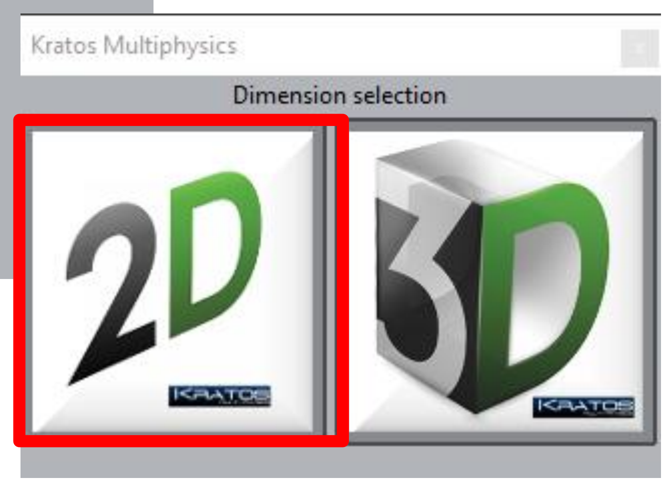


Predefined FSI example

Problem Type

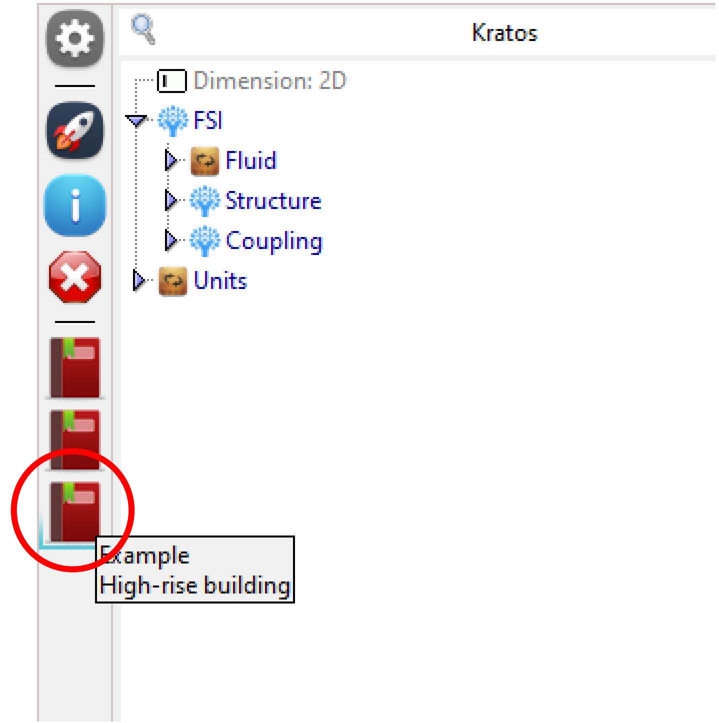
- Load the Kratos problem type
  Data → Problem type → Kratos
- Select *FSI* in the first window (Application Type) and click the *Next* button
- Select *2D* in the second window (Analysis Type) and click the *Next* button





Predefined example „High-rise building“

- Load the predefined example “High-rise building”



- Note the new list of properties for FSI
  - Fluid
  - Structure
  - Coupling

Fluid properties

- **Fluid**
  - Analysis Type
    - Solution Strategy: Navier Stokes - Fractional Step
    - Scheme: Fractional step generic scheme
  - **Parts**
    - group: Fluid
  - Initial Conditions
  - **Conditions**
    - **Automatic inlet velocity**
      - group: Inlet//inlet1
      - group: Inlet//inlet2
    - **Outlet pressure**
      - group: Outlet
    - **Slip**
      - group: Top_Wall
      - group: Bottom_Wall
    - No Slip
    - Custom velocity constraints
    - **ALE mesh displacement BC**
      - group: FluidALEMeshBC//Total
    - **FSI fluid interface**
      - group: InterfaceFluid

- Solution
  - Strategy parameters
    - Echo level: 1
    - Compute reactions: Yes
    - Dynamic tau: 0.1
    - Predictor corrector: No
    - Velocity tolerance: 1e-3
    - Maximum velocity iterations: 15
  - Gravity
  - Time parameters
    - Start time: 0.0
    - End time: 25
    - Automatic delta time: No
    - Delta time: 0.05
  - Velocity linear solver
    - Solver: AMGCL
    - Scaling: No
    - Max iterations: 200
    - Tolerance: 1e-6
    - Krylov space dimension: 100
    - Provide coordinates: No
    - Smoother type: ILU0
    - Krylov type: LGMRES
    - Coarsening type: Aggregation
    - Use block matrices if possible: No
  - Pressure linear solver
    - Solver: AMGCL
    - Scaling: No
    - Max iterations: 200
    - Tolerance: 1e-6
    - Krylov space dimension: 100
    - Provide coordinates: No
    - Smoother type: ILU0
    - Krylov type: LGMRES
    - Coarsening type: Aggregation
    - Use block matrices if possible: No

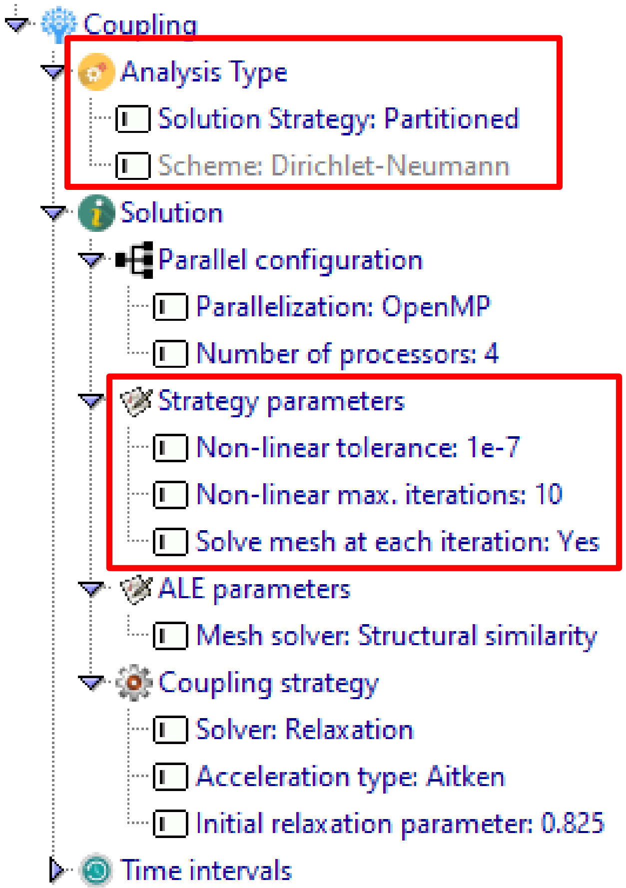
- **Results**
  - On element
  - On nodes
  - **GiD-Output**
    - Enable output: Yes
    - Options
    - Cut planes
  - **Drag**
    - group: InterfaceFluid
      - Write drag output file: 1
      - Print drag to screen: 0
      - Time interval: Total
  - Vtk-Output

Structure properties

- **Structure**
  - Model information
    - Solver Type: Dynamic
    - Time integration method: Implicit
    - Scheme: Bossak
    - Analysis type: Nonlinear
  - **Parts**
    - **Solid**
      - group: Structure
        - Element: Solid total lagrangian
        - Constitutive law: Linear Elastic Plane Stress
        - Material: Steel
        - Density: 7850 kg/m^3
        - Young Modulus: 206.9e9 Pa
        - Poisson Ratio: 0.29
        - Thickness: 0.1 m
    - Beam
    - Truss
    - Cable
  - **Boundary conditions**
    - **Displacement**
      - group: Ground
        - Component X: By value
        - Value X: 0.0 m
        - Component Y: By value
        - Value Y: 0.0 m
        - Time interval: Total

- Solution
  - Strategy parameters
    - Echo level: 1
    - Rayleigh alpha: 0.0
    - Rayleigh beta: 0.0
    - Line search: No
    - Convergence criterion: Displacement criterion
    - Displacement relative tolerance: 1e-4
    - Displacement absolute tolerance: 1e-9
    - Maximum iterations: 10
  - Time parameters
    - Start time: 0.0
    - End time: 25
    - Delta time: 0.05
  - Linear Solver
    - Solver: Automatic

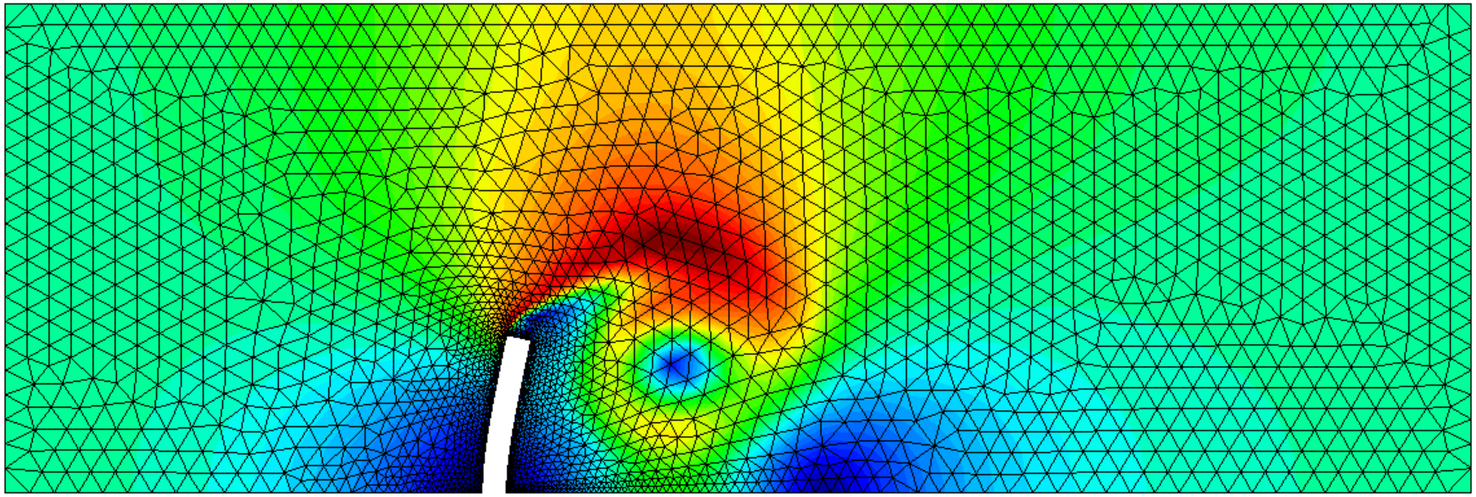
Coupling properties



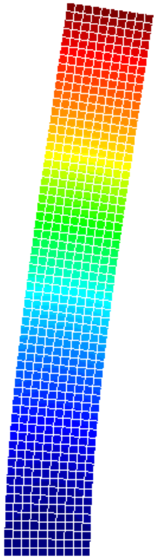
Solution Postprocessing

Results

- The results for the fluid and for the structure are saved in separate files. (file ending: … *.post.bin*)
- Take a look at the results of the velocity of the fluid and the deformation of the structure (select an appropriate displacement factor)
- Observe the influence of the fluid on the structure and vice versa.



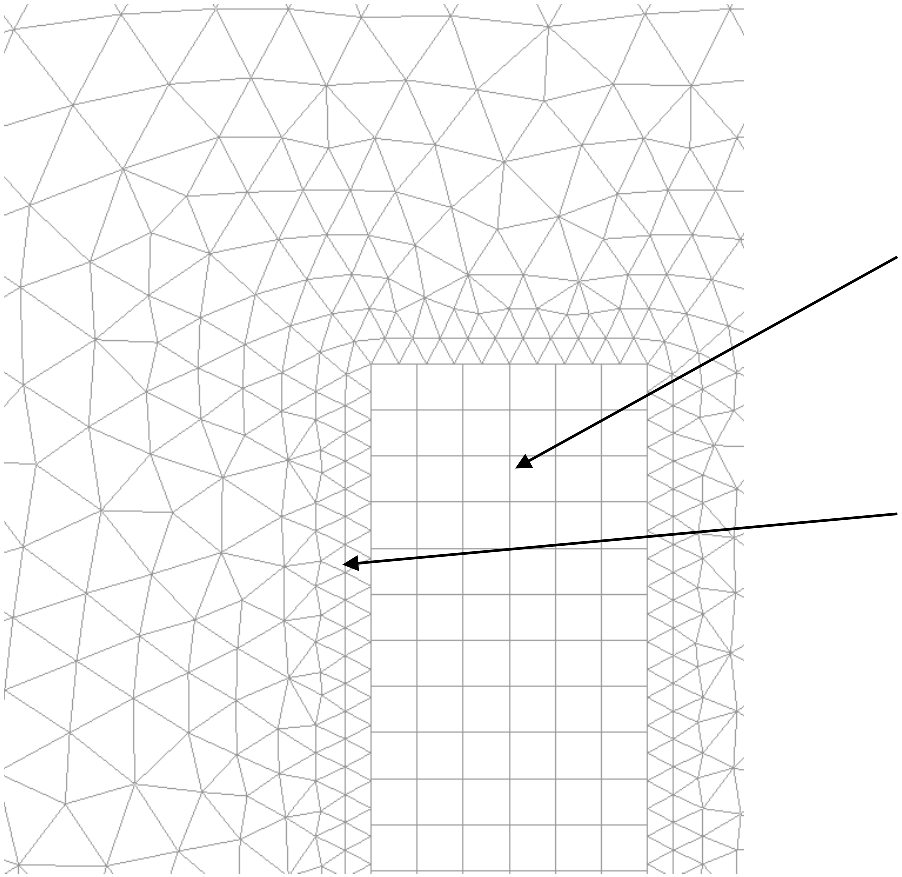
Velocity field
in the fluid domain



Displacement field
in the structural domain

Some theoretical aspects of a FSI simulation

(Non-matching) Grid Mapping



Structural domain:
quadrilateral 2D-plain strain elements

Fluid domain:
2D-triangles

(Non-matching) Grid Mapping

- The building is discretized with quadrilateral 2D-plain solid elements. The fluid domain is discretized with 2D-triangles.
- Most of the times the fluid domain is finer discretized than the structural domain. Then the nodes on the interface are not matching.
- This leads to the requirement that information has to be transferred between the two meshes without the nodes being in the same geometrical position.
- Take a look at what these functions do and connect it to the theory from the lecture.
- The main functions are:
  - *NeumannToStructureNonMatch* → Assigns the reaction forces from the fluid to the structure
  - *DisplacementToMeshNonMatch* → Assigns the displacements of the structure to the fluid mesh
  - Also before the calculation can be started, a coupling matrix for mapping between fluid and structural mesh has to be set up

Structural Solver

- The structural solver is responsible for performing the structural simulation.
- Kratos uses a solver based on the Finite-Element-Method (FEM).
- The simulation can be carried out in many different ways, i.e. considering only a linear behavior or inluding nonlinear phenomena. It is refered to other courses for Finite-Elements.
- The typical FSI simulation is conducted dynamically by considering large displacements, i.e. kinematic nonlinearities.
- Boundary Conditions for the structural simulation:
  - Reaction forces from fluid solver

Fluid Solver

- The fluid solver is responsible for performing the fluid simulation.
- As for the fluid solver, Kratos uses a FEM based solver. This is different from many other solvers, as the most common method to solve fluid problems is the Finite-Volume-Method.
- In the general case, the Navier-Stokes equations are solved. Due to the convection term, these equations are always nonlinear.
- Therefore and because the fluid domain is normally finer discretized, the fluid part in a FSI simulation is the most expensive part in terms of computational effort.
- Boundary Conditions for the fluid simulation:
  - Mesh velocity from mesh solver
  - Changing Domain due to moving interface

Monolithic - Staggered

Another very important aspect of FSI-simulations is the order of executing the different simulations.

Two major approaches exist:

- Monolithic approach: All simulations are conducted at the same time. Instead of having a system of equation for every simulation, all systems are combinded in one big system of equations. This approach is computationally more expensive but more stable.

- Partitioned approach: The simulations are conducted successively. The effort of formulating the systems of equations is low because the formulations for the simulations are already established. However this approach is less stable and often harder to converge. This approach is used for our simulations

Coupling Fluid & Structure

The coupling between the structure and the fluid for the used staggered approach is called a Dirichlet-Neumann coupling scheme.

This refers to the boundary conditions used by solvers that were mentioned above.

There are two ways of coupling:

- Weak coupling (staggered): The three fields (mesh, fluid & structure) are solved once per timestep. Uses a prediction.
- Strong coupling: The fluids are solved several times until the interface displacements are converged in an iterative way. This coupling is used for our simulations.

Relaxation

- The following information is taken from this paper:
- *Fixed-point fluid-structure interaction solvers with dynamic relaxation* by Ulrich Küttler and Wolfgang A. Wall

- In order to ensure and accelerate convergence of the iteration a relaxation step is needed after each FSI cycle.

$$\boldsymbol{d}_{\Gamma,i+1} = \boldsymbol{d}_{\Gamma,i} + \ \omega_i * \boldsymbol{r}_{\Gamma,i+1}$$

- This is necessary for the interface displacements.

- $\boldsymbol{d}_{\boldsymbol{\Gamma}}$ are the interface displacements, $\boldsymbol{\omega}$ is the relaxation parameter, $\boldsymbol{r}$ is the interface residual and $\boldsymbol{i}$ is the iteration number of the strong coupling.

- The relaxation parameter can be kept fixed for the whole simulation or recalculated in every iteration using Aitken‘s method for an optimal convergence.