

Assignment 3: option CFX – flow around an elastically mounted cylinder

- The assignment should be handed in at the latest Wednesday 27th March 2024 12:00 in order to receive full credit, otherwise a 2 point penalty is applied to the grade of the assignment.
- The assignment can be made in groups of **at most 3 students**.
- Please be sure to include the names and student numbers in your report.
- Please hand in an electronic version (also a scanned version of a (clearly) hand-written report is accepted) through Brightspace (only one submission per group is required)
- You do not have to make a full report (including summary, list of figures, list of symbols etc), but indicate clearly the question numbers when giving your answer / result / discussion of results.

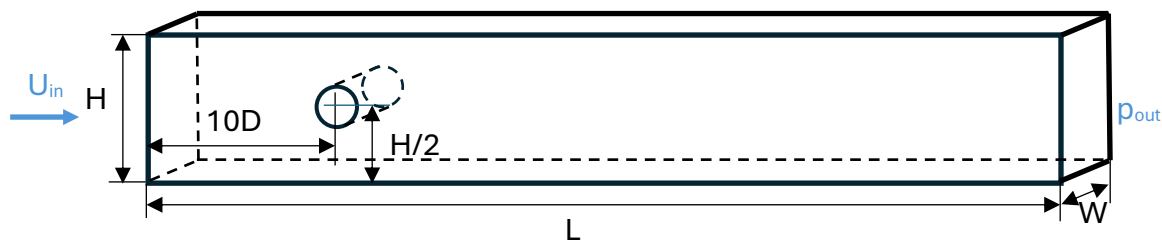
Introduction

In this assignment you will explore the capabilities of Ansys/CFX to model rigid-body FSI motion. As a test case we consider the flow around an elastically mounted cylinder that will operate within the so-called lock-in region. This is the region where the vortex shedding of the flow and the vibration of the structure are synchronized. The purpose of the assignment is to use this testcase and comment on the different options/settings available in CFX and connect those options to the theoretical content given in the lectures.

Please also refer to the “Tutorial: Flow around a circular cylinder using CFX” and “Rigid body motion coupling for the piston problem” on Brightspace for more information about the use of CFX.

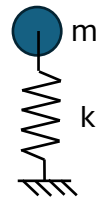
The conditions and the mesh/domain for the flow are exactly the same as the one in the “Flow around a circular cylinder using CFX” tutorial.

The size of the domain, expressed in the cylinder diameter D is: $H = 10D$, $L = 60D$, $W = 1D$. The center of the cylinder is located in the middle of the domain vertically, and located $10D$ from the inlet.



We choose the diameter based Reynolds number to be $Re = \frac{\rho U D}{\mu} = 200$. Here, ρ denotes the fluid density and μ the dynamic viscosity. The flow is considered to be air at 25 degree Celcius. At the inlet, the flow is entering with a uniform velocity $U = 0.03$ [m/s] and at the exit a static relative pressure $p_{out} = 0$ [Pa] is imposed. The top and bottom walls of the channel as assumed to be slip-walls (i.e. only the velocity normal to the wall is zero, and the tangential velocity is free – there is no wall shear stress). The cylinder wall has a no-slip condition (i.e. the velocity of the flow must be equal to zero in both normal and tangential direction).

For the elastically suspended cylinder, we choose a diameter $D = 0.1$ [m], mass $m = 0.001$ [kg] and spring stiffness $k = 1.42e-4$ [N/m].



Exercise 1.1 To get some interpretation of the relation between the fluid and structure parameters, compute the density ratio of the fluid compared to the structure. Also, assume a Strouhal number $St = \frac{fD}{U} = 0.2$ and compute the ratio of the vortex shedding frequency to natural frequency of the structure.

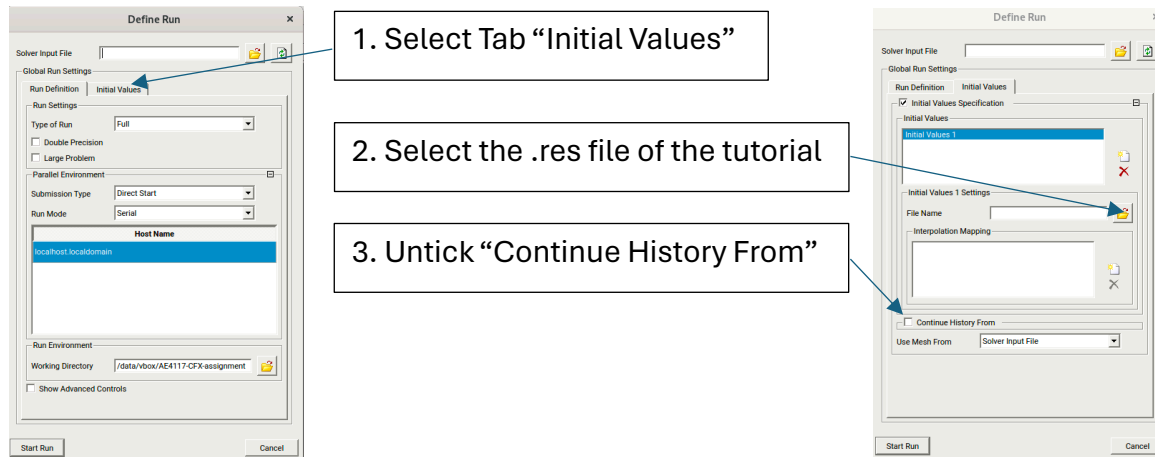
Starting from the settings that you already have for solving the flow around the static cylinder, in order to run the FSI case, you will need to add/modify the following components to the testcase:

- Rigid body
- Mesh deformation
- Boundary conditions
- Solver Control

Exercise 1.2 For each of these settings, discuss what you need to change. Indicate how the kinematic and dynamics boundary conditions are imposed, whether mesh deformation is using absolute or relative displacements, whether a loosely or strongly coupled algorithm is used.

After setting up these conditions, you can save your .def file and continue to run the simulation. We can use the solution from the “Tutorial: Flow around a circular cylinder using CFX” as initial condition. This way, the flow is already developed into a vortex-shedding

state, and then the cylinder is “released” to start its vibration. In the CFX-Solver Manager, you can specify the use of an initial condition from a previous run:



Exercise 2.1 Run simulations with the following settings: simulation time 200 [s], use the time step 0.4 [s] and 0.8 [s]. Run a simulation using a strongly coupled approach and a loosely coupled approach. In total you will have 4 simulations. Present the solution in a y-t diagram (y displacement versus time) and discuss the results.

Exercise 2.2 To analyze the work transfer between the flow and the structure, plot the results of the above 4 simulations in a F_y -y diagram (Resultant fluid force in y-direction versus y displacement). Explain how this diagram can indicate energy transfer from flow to structure. What would you expect in the diagram when the vibration motion is at constant amplitude?

Exercise 3.1 Run one simulation using absolute mesh motion and one simulation using relative mesh motion – visualize and discuss the impact on the mesh.