FLUID-STRUCTURE INTERACTION OF ELASTIC CYLINDRICAL STRUCTURES WITH MODERATELY HIGH TURBULENT FLOW

Carsten Corte
International Center for Numerical Methods in Engineering (CIMNE)
Universidad Politécnica de Cataluña
Campus Norte UPC, 08034 Barcelona, Spain
e-mail: corte@cimne.upc.edu, web page: http://www.cimne.com

Key words: Finite calculus approach (FIC), fluid-structure interaction, slender structures

Abstract. Slender flexible structures can be exposed to excitation by surrounding fluid flow. Long-term excitation can therefore lead to material fatigue and eventually to complete failure of the structure. Numerical simulation to model fluid-structure interaction is an innovative approach to assess the dynamics of slender structures. Applying the finite calculus approach (FIC) to solve the Navier-Stokes equations numerically and a standard finite element approach (FEM) to solve the structural equation of motion, a coupling scheme to solve the coupled problem is presented. To account for the subsequent mesh deformation of the fluid domain mesh, an arbitrary Lagrangean-Eulerian method (ALE) is used. With the coupled approach fluid flow around an elastic thin-wall cylindrical structure is modelled, using an implicit stabilization in case of moderately high Reynolds number flow. The GiD pre- and postprocessing software is applied to set up a joint finite element model for combined computation of fluid-structure interaction using the Tdyn code and the Ramseries code of Compass Ingeniería y Sistemas.

1 INTRODUCTION

The finite calculus approach (FIC) to solve the Navier-Stokes equations has been presented by Oñate, García et al. with application to a variety of engineering problems. An approach for the stabilization of the convective parts of the governing equations has been proposed by García et al. Standard methods for the time integration for the numerical solution of the structural equation of motion are based on the Newmark scheme. Simulation of fluid-structure interaction can be obtained by coupling above methods by mutual data interchange between the fluid and the structural solver scheme.

2 GOVERNING EQUATIONS

2.1 Fluid flow equations

Viscous flow is modelled by the Navier-Stokes equations. To obtain a numerical solution,
the finite calculus approach (FIC) is applied.

2.2 Structural equation of motion

The dynamic motion of a structure can be modelled by the structural equation of motion. To evaluate the transient motion of an elastic structure, a standard finite element discretization is applied in combination with a Newmark family time integration scheme.

2.3 Coupling scheme, ALE formulation

To couple the fluid and the structural solution of the respective governing equations, a coupling scheme to interchange data at the common fluid-structure interface is applied. Further, to consider a moving fluid domain due to the deformation of the structure at the fluid-structure interface, an arbitrary Langrangean-Eulerian approach (ALE) is used. The coupled scheme is split into eight subsequent steps:

1. Solve the fluid equations in the current deformed fluid domain for pressure and velocity at time step \( t_n \) iteratively, thereby considering the current mesh movement velocity of each node in the fluid domain.
2. Determine the current fluid stress vector at each fluid node on the fluid-structure interface at time step \( t_n \).
3. Interpolate the fluid stress vector from the fluid nodes on the fluid-structure interface onto the structural nodes on the fluid-structure interface. For non-matching meshes for the fluid and the structure on the fluid-structure interface, linear interpolation of the fluid stress vector within a fluid element is performed to obtain an interpolated value of the fluid stress vector at the location of the considered structural node.
4. Compute the structural displacement due to the nodewise applied fluid stress vector onto each structural node on the fluid-structure interface at time step \( t_n \).
5. Interpolate the structural displacement increment (the difference between nodal structural displacement at time step \( t_n \) and at time step \( t_{n-1} \)) from the structural nodes on the fluid-structure interface onto the fluid nodes on the fluid-structure interface. For non-matching meshes for the fluid and the structure on the fluid-structure interface, linear interpolation of the structural displacement increment within a structural element is performed to obtain an interpolated value of the structural displacement increment at the location of the considered fluid node.
6. Compute the fluid domain mesh deformation due to structural boundary deformation of the fluid-structure interface at time step \( t_n \). Therefore, the following concept is applied: As boundary condition for the fluid domain mesh deformation, the deformation increment on the fluid-structure interface of the fluid domain mesh (FSI boundary) is given by the interpolated structural displacement increments. On another part of the boundary of the fluid domain mesh, a zero displacement boundary condition for the fluid domain mesh movement is imposed (fixed boundary). On the rest of the boundary of the fluid domain mesh, fluid nodes are left to move according to the fluid domain mesh deformation procedure. The fluid domain mesh deformation procedure is illustrated separately in the paragraph below.
7. Update the current fluid mesh velocity nodewise by dividing the nodewise fluid mesh deformation increment by the current time step increment.
8. Increment time index \( n \leftarrow n+1 \) and go to step 1.
The deformation of the fluid nodes between the FSI boundary and the fixed boundary of the fluid domain mesh is determined by an interpolation procedure. For the interpolation procedure, a binary tree approach is applied. In the binary tree, the fluid nodes are ordered in different layers. The top layer consists of the nodes that belong to the FSI boundary of the fluid domain mesh. The next layer consists of fluid nodes that are adjacent to the FSI boundary nodes. This setup of the binary tree is performed until all fluid nodes are positioned in the binary tree. The deformation increment for a fluid node in a certain layer is preliminarily obtained by the average mesh deformation increment of all nodes in the layer above that are connected to the considered node by the binary tree. This procedure gives a preliminary deformation increment for each fluid node. Subsequently, a nodewise specific scaling factor with value between zero and one is applied to the preliminary deformation increment of each fluid node. The scaling factor depends on the distance of the considered fluid node to the FSI boundary of the fluid domain mesh. For fluid nodes on the FSI boundary, the scaling factor is one; for fluid nodes on the fixed boundary, the scaling factor is zero. For fluid nodes between the FSI boundary and the fixed boundary, the scaling factor is determined by a nonlinear function, depending on the distance of the considered fluid node from the FSI boundary. The nodally determined scaling factor is applied to the nodewise preliminary deformation increment to obtain the mesh deformation increment of each fluid node. The obtained nodal mesh deformation increment is further smoothened over the fluid domain mesh, regarding the fluid element size (element volume) distribution over the fluid domain. Finally, the mesh deformation increment is added nodewise to the fluid mesh node coordinates.

3 COMPUTATIONAL EXAMPLE

The coupled approach has been applied to model the structural response of a thin-wall elastic cylinder due to surrounding fluid flow of Reynolds number 10000. Geometric data of the cylinder was height 8 m, diameter 2 m and wall thickness 0.01 m. The cylinder was fixed at the bottom and free at its top. The cylinder had isotropic material properties of modulus of elasticity $E = 2.1e8 \text{ N/m}^2$, density $\rho = 7860 \text{ kg/m}^3$ and Poisson’s ratio $\nu = 0.3$. The fluid material properties were density $\rho_F = 1 \text{ kg/m}^3$ and viscosity $\nu_F = 0.0002 \text{ kg/(m.s)}$. The inflow velocity was constant with 1 m/s in longitudinal flow direction.

Figure 1: User interface for FSI cylinder model  
Figure 3: Pressure distribution
Figure 2: Deformed cylinder excited by fluid flow
Figure 4: Velocity distribution

In figure 3 and 4 the pressure and velocity distribution around the cylinder are shown. Vortex evolution in the rear of the cylinder can clearly be identified. The vortex shedding frequency corresponds well with the Strouhal number of 0.2 at the considered Reynolds number. Figure 2 shows the deformed cylindrical structure as response to fluid flow excitation. It is clearly shown that the thin-wall cylinder does not behave as a beam-like structure. There is clear ovaling of the cylinder shell, which is in well accordance with the eigenforms of the considered structure.

4 CONCLUSIONS

The presented coupling scheme to simulate fluid-structure interaction between the finite calculus approach (FIC) solution of the Navier-Stokes equations and a standard finite element solution of the structural equation of motion shows adequate results for the case of moderately high Reynolds number flow around a thin-wall elastic cylindrical structure. The structural response corresponds to the fluid flow excitation pattern. Moderately large structural deformations at the fluid-structure interface can be handled by the mesh deformation scheme for the fluid domain mesh.

REFERENCES
