

# Contents

<b>1</b>	<b>Verification and Validation</b>	<b>3</b>
1.1	Verification of Code . . . . .	3
1.1.1	Method of manufactured solution . . . . .	4
1.1.2	Comment on verification of the fluid-structure interaction solver by MMS . . . . .	5
1.2	Validation of Code . . . . .	6
1.2.1	Validation of fluid solver . . . . .	8
1.2.2	Validation of solid solver . . . . .	12
1.2.3	Validation of fluid structure interaction solver . . . . .	16



# A motivation for studying fluid-structure interaction

Fluid-structure interaction(FSI) is an interdisciplinary field, appearing in many applications. In nature, FSI forms the basis of many physical phenomena. A fish swimming upstream, generating thrust from the surrounding fluid by wave-like movements of its fin and body. Or a tree, bending back and forth due to strong winds of a storm passing by. Both examples are understandable, but points out two main instances of how FSI occur. When the fish swims, it deforms the fluid, altering the nearby flowfield. For the tree however, the swinging and bending is induced by the pressure of passing wind acting on the tree trunk and branches. Ultimately, fluid-structure interaction occurs due to both initial effect of either fluid, structure or a combination.

---

Computational fluid-structure interaction (CFSI) has grown vast within engineering in the recent years, and proved to be essential for design development and performance optimization of many applications. Applications are, but not limited to biomedical computations such as heart valves and aneurisms([?], [?]) inflation of parachutes [?], Underwater explosions [?] and wind turbines [?].

Within aeronautics, CFSI have proven to be crucial for advances within flight characteristics and fuel economy. Due to a wide range of wing materials and flow profiles to be studied, CFSI have made testing of proposed models possible, while saving expenses regarding small and full-scale experiments.

Winglet, a near vertical tip replacement for a conventional wingtip of an aircraft, have reduced drag induced by wingtip vortices during flight. As a result, the overall fuel consumption of long-distance flights have been reduced by  $\sim 5\%$ , which is why winglets can be observed within many airliners today. Another consequence of installing winglets is the reduction of wingtip vortices, which in turn reduces trailing turbulence behind the aircraft. The trailing turbulence can intervene with flight controls of aircraft passing through it, making winglets an important safety feature for flight traffic.



Figure 1.1: A comparison of shedding vortices from conventional wingtip, versus a winglet.

Given the multidisciplinary nature of FSI, significant developments within the field have occurred within recent years. Traditionally, fluid and structure mechanics have been considered separate scientific fields, however the complex interaction between them. There are several causes for this, coupling of equations and nonlinearity.

Even though FSI play an important role within many scientific applications, Computational stuff, why now, refer to computational power etc. Få med beregningsorienterte, to interdisciplinary.

# Governing equations

Computational fluid-structure interaction (CFSI) combines two separate fields of computational mechanics, computational fluid dynamics (CFD), and computational structure dynamics (CSM). While CFD and CSM traditionally have been considered as two distinct fields of science, the goal of CFSI is to combine the separate fluid and structure problems, and their interaction or *coupling* to one another. Therefore, the study CFSI demands understanding of each separate field. This chapter presents the governing equations of the individual fluid and structure problem. Balance laws for each separate problem, together with auxiliary kinematic, dynamic and material relations will be described briefly.

## 2.1 Continuum Mechanics

In our effort to understand and describe physical phenomenon in nature, we describe our observations and theories by creating mathematical models. The mathematical models makes scientist and engineers not only able to understand physical phenomena, but also predict them. All matter is built up by a sequence of atoms, meaning on a microscopic level, an observer will locate discontinuities and space within the material. Evaluating each atom, or *material point*, is not impossible from a mathematical point of view. However, for mathematical modeling and applications, the evaluation of each material point remains unpractical. In *continuum mechanics*, first formulated by Augustin-Louis Cauchy [?], the microscopic structure of materials are ignored, assuming the material of interest is *continuously distributed* in space, referred to as a continuum.

In context of this thesis we define a *continuum* as a continuous body  $V(t) \subset \mathbb{R}^d$   $d \in (1, 2, 3)$ , continuously distributed throughout its own extension. The continuum is assumed to be infinitely divisible, meaning one can divide some region of the continuum a indefinitely number of times. A continuum is also assumed to be locally homogeneous, meaning if a continuum is subdivided into infinitesimal regions, they would all have the same properties such as mass density. These two properties forms the baseline for deriving conservation laws and constitute equations, which are essential for formulating mathematical models for the material of interest. However, a continuum remains a mathematical idealization, and may not be a reasonable model for certain applications. In general, continuum mechanics have proven to be applicable provided that  $\frac{\delta}{l} \ll 1$  where  $\delta$  is a characteristic length scale of the material micro-structure, and  $l$  is a length scale of the problem of interest [?].

## 2.2 The Lagrangian and Eulerian description of motion

In continuum mechanics, one makes use of two classical descriptions of motion, the *Lagrangian* and *Eulerian* description. Both concepts are related to an observer's view of motion, visually explained by the concepts of *material* and *spatial* points. A material point represents a particle within the material, moving with the material as it moves and deforms. A spatial point, refers to some reference at which the path of the material points are measured from.

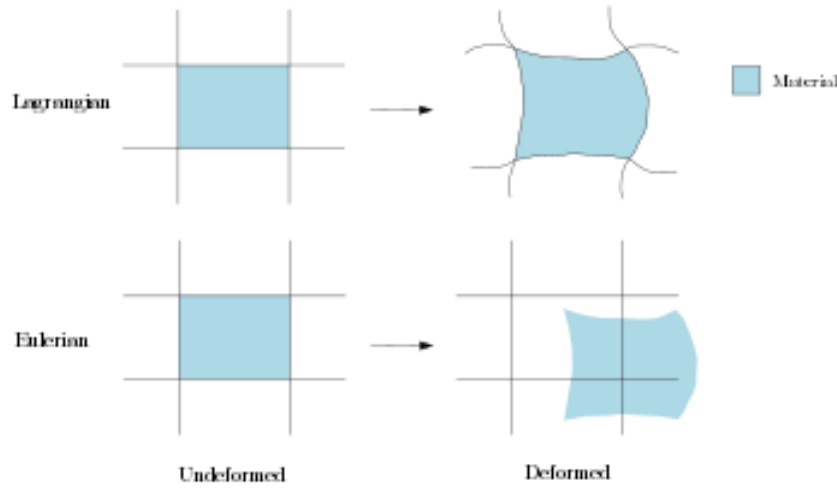


Figure 2.1: Comparison of the Lagrangian and Eulerian description of motion

### Lagrangian

In the Lagrangian description of motion, the material and spatial points coincide, meaning the reference point of which motion is measured, follows the material as it deviates from its initial position. The initial position of all material points in a continuum extend a region, called the *reference configuration*  $\hat{V}$ . From now on, all identities in the *reference configuration* will be denoted with the notation " $\wedge$ ". If a continuum deviates from its reference configuration, a material point  $\hat{\mathbf{x}}(x, y, z, t)$  may no longer be at its initial position, but moved to a new position  $\mathbf{x}(x, y, z, t)$  at time  $t$ . The new positions of all material points extend a new region, called the *current configuration*  $V(t)$ .

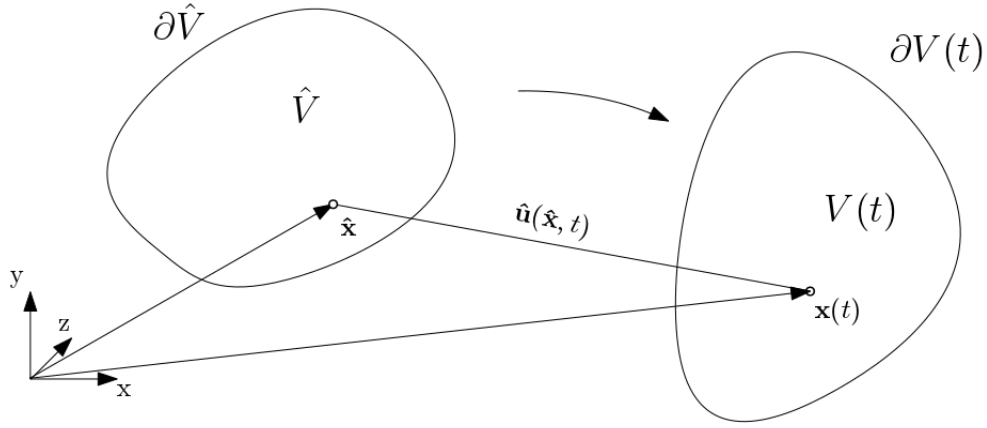


Figure 2.2: Unitpotato

To measure the displacement of a material point  $\mathbf{x} \in V(t)$  for time  $t$ , from its initial point  $\hat{\mathbf{x}} \in \hat{V}$ , one defines a *deformation vector field*

$$\hat{\mathbf{u}}(\hat{\mathbf{x}}, t) = \mathbf{x}(\hat{\mathbf{x}}, t) - \hat{\mathbf{x}} = \hat{\mathbf{T}}(\hat{\mathbf{x}}, t) \quad (2.1)$$

Mathematically, deformation is a 1:1 mapping  $\hat{\mathbf{T}}(\hat{\mathbf{x}}, t)$ , transforming material points from the *reference configuration*  $\hat{V}$ , to the *current configuration*  $V(t)$ . Visually, the deformation resembles the shape of continuum for some time  $t$ . To describe the continuums motion, one defines the *velocity vector field* given by the time derivative of the deformation field,

$$\hat{\mathbf{v}}(\hat{\mathbf{x}}, t) = d_t \mathbf{x}(\hat{\mathbf{x}}, t) = d_t \hat{\mathbf{u}}(\hat{\mathbf{x}}, t) = \frac{\partial \hat{\mathbf{T}}(\hat{\mathbf{x}}, t)}{\partial t} \quad (2.2)$$

The Lagrangian description of motion is the natural choice when tracking particles and surfaces are of main interest. Therefore, it is mainly used within structure mechanics.

## Eulerian

In the Eulerian description, the reference point is kept fixed while the material is deformed.

Considering a flow of fluid particles in a river, a *Lagrangian* description of the particles would be tedious as the number of particles entering and leaving the domain quickly rise to a immense number. Instead consider defining a view-point  $V$  fixed in time, and monitor every fluid particle passing the coordinate  $x \in V(t)$  as time elapses. Such a description is defined as the *Eulerian framework*. Therefore the Eulerian formulation is natural for describing fluid dynamics.

We can describe the particles occupying the *current configuration*  $V(t)$  for some time  $t \geq t_0$

$$x = \hat{\mathbf{x}} + \hat{\mathbf{u}}(\hat{\mathbf{x}}, t)$$

Since our domain is fixed we can define the deformation for a particle occupying position  $x = x(\hat{x}, t)$  as

$$\mathbf{u}(x, t) = \hat{u}(\hat{x}, t) = x - \hat{x}$$

and its velocity

$$\mathbf{v}(x, t) = \partial_t u(x, t) = \partial_t \hat{u}(\hat{x}, t) = \hat{v}(\hat{x}, t)$$

It is important to mention that we are not interested in which particle is occupying a certain point in our domain, but only its properties.

## 2.3 The deformation gradient

Deformation is a major property of interest when a continuum is influenced by external and internal forces. The deformation results in relative change of position of material particles, called *strain*. and is the primary property that causes and describe *stress*.

Strain is purely an observation, and it is not dependent on the material of interest. However one expects that a material undergoing strain, will give forces within a continuum due to neighboring material particles interacting with one another. Therefore one derive material specific models to describe how a certain material will react to a certain amount of strain. These strain measures are used to define models for *stress*, which is responsible for the deformation in materials [?]. Stress is defined as the internal forces that particles within a continuous material exert on each other, with dimension force per unit area.

The equations of continuum mechanics can be derived with respect to either a deformed or undeformed configuration. The choice of referring our equations to the current or reference configuration is indifferent from a theoretical point of view. In practice however this choice can have a severe impact on our strategy of solution methods and physical of modelling. [?]. Regardless of configuration, the *deformation gradient* and *determinant of the deformation gradient* are essential measurement in structure mechanics. By [?], both configurations are considered.

### Reference configuration

**Definition 2.1.** Let  $\hat{u}$  be a differential deformation field in the *reference* configuration,  $I$  be the Identity matrix and the gradient  $\hat{\nabla} = (\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z})$ . Then the *deformation gradient* is given by,

$$\hat{\mathbf{F}} = I + \hat{\nabla} \hat{u} \tag{2.3}$$

*expressing the local change of relative position under deformation.*



**Definition 2.2.** Let  $\hat{\mathbf{u}}$  be a differential deformation field in the *reference* configuration,  $I$  be the Identity matrix and the gradient  $\hat{\nabla} = (\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z})$ . Then the *determinant of the deformation gradient* is given by,

$$J = \det(\hat{\mathbf{F}}) = \det(I + \hat{\nabla} \hat{\mathbf{u}}) \quad (2.4)$$

*expressing the local change of volume the configuration.*

From the assumption of linear operator  $\mathbf{F}$ , and no two particles  $\hat{\mathbf{x}}_a, \hat{\mathbf{x}}_b \in \hat{V}$  occupy the same location for some time  $V(t)$ ,  $J$  to be greater than 0 [?].

### Current configuration

**Definition 2.3.** Let  $\mathbf{u}$  be a differential deformation field in the *reference* configuration,  $I$  be the Identity matrix and the gradient  $\nabla = (\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z})$ . Then the *deformation gradient* is given by,

$$\mathbf{F} = I - \nabla \mathbf{u} \quad (2.5)$$

*expressing the local change of relative position under deformation.*

**Definition 2.4.** Let  $\mathbf{u}$  be a differential deformation field in the *reference* configuration,  $I$  be the Identity matrix and the gradient  $\nabla = (\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z})$ . Then the *determinant of the deformation gradient* is given by,

$$J = \det(\mathbf{F}) = \det(I - \nabla \mathbf{u}) \quad (2.6)$$

*expressing the local change of volume the configuration.*

## 2.4 The solid problem

The governing equations for the solid mechanics are given by the balance law,

**Equation 2.4.1.** *Solid momentum*

$$\rho_s \frac{\partial \hat{\mathbf{v}}_s}{\partial t} = \nabla \cdot \hat{\mathbf{S}} + \rho_s \mathbf{f}_s \quad \text{in } \hat{\Omega}_s \quad (2.7)$$

$$\frac{\partial \hat{\mathbf{v}}_s}{\partial t} = \hat{\mathbf{u}}_s \quad \text{in } \hat{\Omega}_s \quad (2.8)$$

defined in a Lagrangian coordinate system, with respect to an initial reference configuration  $\hat{\Omega}_s$ . The structure configuration is given by the displacement  $\hat{\mathbf{u}}_s$ , with the relation  $\frac{\partial \hat{\mathbf{v}}_s}{\partial t} = \hat{\mathbf{u}}_s$  to the solid velocity. The density of the structure is given by  $\rho_s$ , and  $\hat{\mathbf{f}}_s$  express any exterior body forces acting. Finally,  $\hat{\mathbf{S}}$  is the second Piola-Kirchhoff stress tensor, related to the Cauchy-stress tensor by,

$$\hat{\mathbf{S}}_s = J \hat{\mathbf{F}}^{-1} \sigma_s \hat{\mathbf{F}}^{-T}$$

The elasticity of the material is expressed by the *Poisson ratio*  $\nu_s$ , *Young modulus*  $E$ , or Lamè coefficients  $\lambda_s$  and  $\mu_s$ . Their relation is given by,

$$\begin{aligned} E_y &= \frac{\mu_s(\lambda_s + 2\mu_s)}{(\lambda_s + \mu_s)} & \nu_s &= \frac{\lambda_s}{2(\lambda_s + \mu_s)} \\ \lambda_s &= \frac{\nu E_y}{(1 + \nu_s)(1 - 2\nu_s)} & \mu_s &= \frac{E_y}{2(1 + \nu_s)} \end{aligned}$$

Material models express the dependency between strain tensors and stress. The validity of material models is often limited by their ability to handle deformation and strain to some extent, before it breaks down or yields nonphysical observations of the material. For small-deformations, *Hooke's law* assumes a linear relation between strain and stress,

**Definition 2.5.** Let  $\hat{u}$  be a differential deformation field in the *reference* configuration,  $I$  be the Identity matrix and the gradient  $\hat{\nabla} = (\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z})$ . *Hooke's law* is then given by,

$$\begin{aligned} \sigma_s &= \frac{1}{J} \hat{\mathbf{F}} (\lambda_s (Tr(\epsilon)I + 2\mu\epsilon)) \hat{\mathbf{F}}^{-T} \\ \hat{\mathbf{S}}_s &= \lambda_s (Tr(\epsilon)I + 2\mu\epsilon) \\ \epsilon &= \frac{1}{2} (\hat{\nabla} \hat{\mathbf{u}} + (\hat{\nabla} \hat{\mathbf{u}})^T) \end{aligned}$$

Hooke's law is however limited to a small-deformation regime, and is not applicable for larger deformations encountered in this thesis. A valid model for larger deformations is the hyper-elastic *St. Venant-Kirchhoff model* (STVK), extending Hooke's law into a non-linear regime.

**Definition 2.6.** Let  $\hat{u}$  be a differential deformation field in the *reference* configuration,  $I$  be the Identity matrix and the gradient  $\hat{\nabla} = (\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z})$ . The *St. Venant-Kirchhoff model* is then given by the relation,

$$\begin{aligned} \sigma_s &= \frac{1}{J} \hat{\mathbf{F}} (\lambda_s (Tr(\hat{\mathbf{E}})I + 2\mu\hat{\mathbf{E}})) \hat{\mathbf{F}}^{-T} \\ \hat{\mathbf{S}}_s &= \lambda_s (Tr(\hat{\mathbf{E}})I + 2\mu\hat{\mathbf{E}}) \\ \hat{\mathbf{E}} &= \frac{1}{2} (\hat{\mathbf{C}} - I) \quad \hat{\mathbf{C}} = \hat{\mathbf{F}} \hat{\mathbf{F}}^{-T} \end{aligned}$$

where  $\hat{\mathbf{C}}$  is the right Cauchy-Green strain tensor and  $\hat{\mathbf{E}}$  is the Green Lagrangian strain tensor <sup>1</sup>

Though STVK can handle large deformations, it is not valid for large strain [?]. However since the strain considered in this thesis are small, it will remain our primary choice of strain-stress relation. STVK describes materials of compressible nature, but it should be mentioned that for large deformation models describing incompressible materials can be considered. Specially the Incompressible Neo-Hooke

---

<sup>1</sup>See appendix A for definition

(INH) model is considered in several publications (see [?], [?]), sharing the same hyperelastic properties as the STVK model. While both models are valid for large deformations, the INH is superior compared to STVK in the sense that it holds for large strains aswell [?].

As for the fluid problem we define Dirichlet and Neumann boundary conditions on the form

$$\begin{aligned} \mathbf{v}_s &= \mathbf{v}_s^D & \text{on } \Gamma_s^D \subset \partial\Omega_s \\ \sigma_s \cdot \mathbf{n} &= \mathbf{g} & \text{on } \Gamma_s^N \subset \partial\Omega_s \end{aligned}$$

## 2.5 The Fluid problem

The fluid is assumed to be express by the in-compressible Navier-Stokes equations,

**Equation 2.5.1.** *Navier-Stokes equation*

$$\rho \frac{\partial \mathbf{v}_f}{\partial t} + \rho \mathbf{v}_f \cdot \nabla \mathbf{v}_f = \nabla \cdot \sigma + \rho \mathbf{f}_f \quad \text{in } \Omega_f \quad (2.9)$$

$$\nabla \cdot \mathbf{v}_f = 0 \quad \text{in } \Omega_f \quad (2.10)$$

defined in an Eulerian description of motion. The fluid density as  $\rho_f$  and fluid viscosity  $\nu_f$  are assumed to be constant in time, and  $\mathbf{f}_s$  represents any body force. The fluid is assumed Newtonian, where *Cauchy stress sensor* follows Hooke's law

$$\sigma = -p_f I + \mu_f (\nabla \mathbf{v}_f + (\nabla \mathbf{v}_f)^T)$$

Additional appropriate boundary conditions are supplemented to the equation for a given problem. The first type of boundary conditions are Dirichlet boundary conditions,

$$\mathbf{v}_f = \mathbf{v}_f^D \quad \text{on } \Gamma_f^D \subset \partial\Omega_f \quad (2.11)$$

The second type of boundary condition are Neumann boundary conditions

$$\sigma_f \cdot \mathbf{n} = \mathbf{g} \quad \text{on } \Gamma_f^N \subset \partial\Omega_f \quad (2.12)$$

# Computational Fluid Structure Interaction

The multi-disciplinary nature of computational fluid-structure interaction , involves addressing issues regarding computational fluid dynamics and computational structure dynamics. In general, CFD and CSM are individually well-studied, in terms of numerical solution strategies. CFSI adds another layer of complexity to the solution process, (1) the *coupling* of the fluid and solid equations , (2) the tracking of *interface* separating the fluid and solid domains. The coupling pose two new conditions at the interface absent from the original fluid and solid conditions, which is *continuity of velocity* and *continuity of stress* at the interface.

$$\mathbf{v}_f = \mathbf{v}_s \quad (3.1)$$

$$\sigma_f \cdot \mathbf{n} = \sigma_s \cdot \mathbf{n} \quad (3.2)$$

The tracking of the interface is a issue, due to the different description of motion used in the fluid and solid problem. If the natural coordinate system are used for the fluid problem and solid problem, namely the eulerian and lagrangian description of motion, the domains doesn't match and the interface. Tracking the interface is aslo essential for fulfilling the interface boundary conditions. As such only one of the domains can be described in its natural coordinate system, while the other domain needs to be defined in some transformed coordinate system.

Fluid-structure interaction problems are formally divided into the *monolithic* and *partitioned* frameworks. In the monolithic framework, the fluid and solid equations together with interface conditions are solved simultaneously. The monolithic approach is *strongly coupled*, meaning the *kinematic* (1.1) and *dynamic*(1.2) interface conditions are met with high accuracy. However, the complexity of solving all the equations simuntainiously and the strong coupling contributes to a stronger non-linear behaviour of the whole system [15]. The complexity also makes monolithic implementations *ad hoc* and less modular, and the nonlinearity makes solution time slow.

In the *partitioned* framework one solves the equations of fluid and structure subsequently. Sovling the fluid and solid problems individually is beneficial, in terms of the wide range of optimized solvers and solution strategies developed for each sub-problem. In fact, solving the fluid and solid separatly was used in the initial efforts in CFSI, due to existing solvers for one or both problems [2]. Therefore,

---

computational efficiency and code reuse is one of the main reasons for choosing the partitioned approach. A major drawback is the methods ability to enforce the *kinematic* (1.1) and *dynamic*(1.2) conditions at each timestep. Therefore partitioned solution strategies are defined as *weakly* coupled. However, by sub-iterations between each sub-problem at each timestep, (1.1) and (1.2) can be enforced with high accuracy, at the cost of increased computational time.

Regardless of framework, CFSI has to cope with a numerical artifact called the "added-mass effect" [?], [?], [?]. The term is not to be confused with added mass found in fluid mechanics, where virtual mass is added to a system due to an accelerating or de-accelerating body moving through a surrounding fluid [?]. Instead, the term is used to describe the numerical instabilities occurring for weakly coupled schemes, in conjunction with *in compressible fluids* and slender structures [?], or where the density of the incompressible fluid is close to the structure. For partitioned solvers, sub-iterations are needed when the "added-mass effect" is strong, but for incompressible flow the restrictions can lead to unconditional instabilities [2]. The *strong coupled* monolithic schemes have proven overcome "added-mass effect" preserving energy balance, at the prize of a highly non-linear system to be solved at each time step [?].

Capturing the interface is matter of its own, regardless of the the monolithic and partitioned frameworks. The scope of interface methods are divided into *interface-tracking* and *interface-capturing* methods.[?]. In the *Interface-tracking* method, the mesh moves to accommodate for the movement of the structure as it deforms the spatial domain occupied by the fluid. As such, the mesh itself "tracks" the fluid-structure interface as the domain undergoes deformation. Interface-capturing yields better control of mesh resolution near the interface, which in turn yeilds better control of this critical area in terms of enforcing the interface conditions. However, moving the mesh-nodes pose potential problems for mesh-entanglements, restricting the possible extent of deformations.

In *interface-capturing* methods one distinguish the fluid and solid domains by some phase variable over a fixed mesh, not resolved by the mesh iteself. This approach is in general not limitied in terms of deformations, but suffers from reduced accuracy at the interface. [?].

Among the multiple approaches within CFSI, the arbitrary Lagrangia-Eulerian methos is chosen for this thesis.

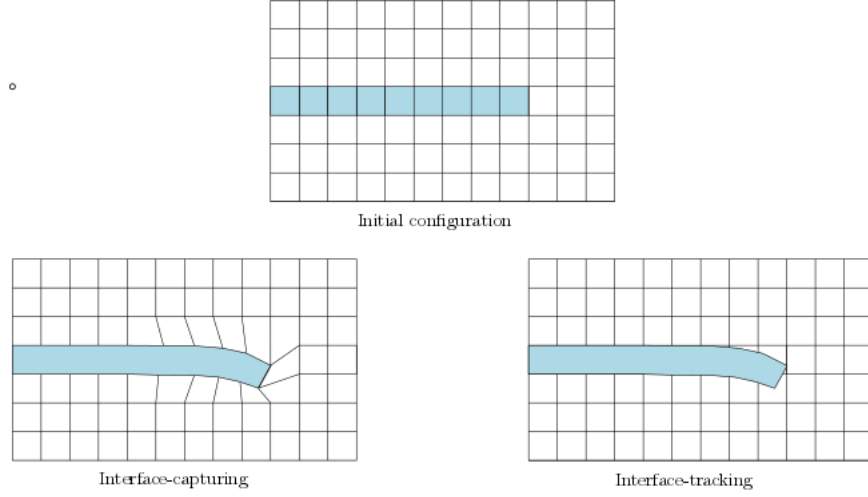


Figure 3.1: Comparison of interface-tracking and interface-capturing for an elastic beam undergoing deformation

### 3.1 Arbitrary Lagrangian Eulerian formulation

The *arbitrary Lagrangian-Eulerian* formulation is the most popular approach within *Interface-tracking* [?], [?]. In this approach the structure is given in its natural *Lagrangian coordinate system*, while the fluid problem is formulated in an artificial coordinate system similar to the *Lagrangian coordinate system*, by an artificial fluid domain map from the underformed *reference configuration*  $\hat{\mathbf{T}}_f(t) : \hat{V}_f(t) \rightarrow V_f(t)$ . The methods consistency is to a large extent dependent on the regularity of the artificial fluid domain map. Loss of regularity can occur for certain domain motions, were the structure makes contact with domain boundaries or self-contact with other structure parts [[?], [?]]. Since no natural displacement occur in the fluid domain, the transformation  $\hat{\mathbf{T}}_f(t)$  has no directly physical meaning [?], [?]. Therefore, the construction of the transformation  $\hat{\mathbf{T}}_f(t)$  is a purely numerical exercise.

### 3.1.1 ALE formulation of the fluid problem

The original fluid problem, defined by the incompressible Navier-Stokes equations (Equation 1.5.1), are defined in an *Eulerian description of motion*  $V_f(t)$ . By changing the computational domain to an underformed *reference configuration*  $V_f(t) \rightarrow \hat{V}_f(t)$ , the original problem no longer comply with the change of coordinate system. Therefore, the original Navier-Stokes equations needs to be transformed onto the *reference configuration*  $\hat{V}_f$ . Introducing the basic properties needed for mapping between the sub-system  $\hat{V}_f(t)$  and  $V_f(t)$ , we will present the ALE time and space derivative transformations found in [?], with help of a new arbitrary fixed reference system  $\hat{W}$

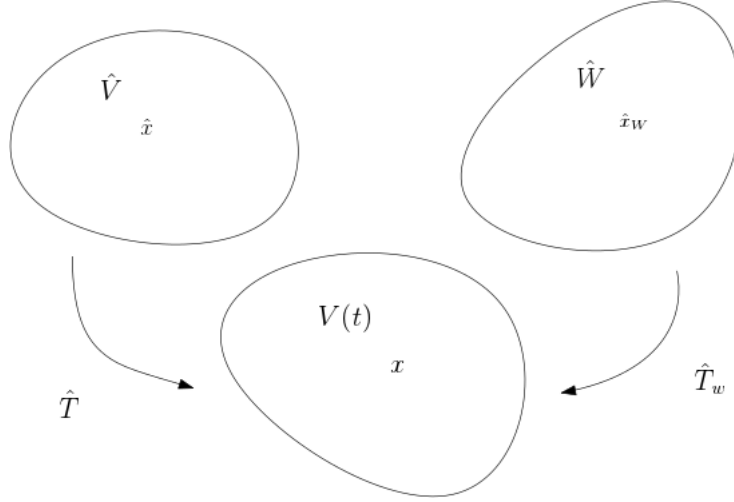


Figure 3.2: CFD-3, flow visualization of velocity time  $t = 9s$

. Let  $\hat{T}_w : \hat{W} \rightarrow V(t)$  be an invertable mapping, with the scalar  $\hat{f}(\hat{x}_w, t) = f(x, t)$  and vector  $\hat{\mathbf{w}}(\hat{x}_w, t) = \mathbf{w}(x, t)$  counterparts. Further let the deformation gradient  $\hat{F}_w$  and its determinant  $\hat{J}_w$ , be defined in accordance with definition 1.1 and 1.2 in Chapter 1. Then the following relations between temporal and spatial derivatives apply, between the two domains  $\hat{W}(t)$  and  $V(t)$ ,

**Lemma 3.1.** Local change of volume

Let  $V(t)$  be the reference configuration  $V(t) \rightarrow \mathbb{R}^d$ , and  $\hat{W} \rightarrow \mathbb{R}^d$  be the arbitrary reference configuration. By the determinant of the deformation gradient  $\hat{J}_w$ , the following relations holds,

$$|V(t)| = \int_{\hat{W}} \hat{J}_w d\hat{x} \quad (3.3)$$

**Lemma 3.2.** Transformation of scalar spatial derivatives

Let  $f$  be a scalar function such that  $f : V(t) \rightarrow \mathbb{R}$ , and  $\nabla f$  be its gradient. Then its counterpart  $\nabla \hat{f}$ , by the scalar function  $\hat{f} : \hat{W} \rightarrow \mathbb{R}$  is given by the relation.

$$\nabla f = \hat{F}_w^{-T} \nabla \hat{f} \quad (3.4)$$



**Lemma 3.3.** *Let  $\mathbf{w}$  be a vector field such that  $\mathbf{w} : V(t) \rightarrow \mathbb{R}^d$ , and  $\nabla \mathbf{w}$  be its gradient. Then its counterpart  $\hat{\nabla} \hat{\mathbf{w}}$ , by the vector field  $\hat{\mathbf{w}} : \hat{W} \rightarrow \mathbb{R}^d$  is given by the relation.*

$$\nabla \mathbf{w} = \hat{\nabla} \hat{\mathbf{w}} \hat{F}_W^{-1} \quad (3.5)$$

**Lemma 3.4.** Transformation of scalar temporal derivatives

*Let  $f$  be a scalar function such that  $f : V(t) \rightarrow \mathbb{R}$ , and  $\frac{\partial f}{\partial t}$  be its time derivative. Then its counterpart  $\frac{\partial \hat{f}}{\partial t}$ , by the scalar function  $\hat{f} : \hat{W} \rightarrow \mathbb{R}$  is given by the relation,*

$$\frac{\partial f}{\partial t} = \frac{\partial \hat{f}}{\partial t} - (\hat{F}_W^{-1} \frac{\partial \hat{\mathbf{T}}_W}{\partial t} \cdot \hat{\nabla}) \hat{f} \quad (3.6)$$

where  $\frac{\partial \hat{\mathbf{T}}_W}{\partial t}$  the domain velocity of  $\hat{W}$

With the necessary preliminaries set, the original fluid problem (Equation 1.1) can be derived with respect to  $\hat{W}$ . By Lemma 2.2, 2.3 the material derivative  $\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v}$  is transformed by,

$$\frac{d\mathbf{v}}{dt} = \frac{\partial \hat{\mathbf{v}}}{\partial t} - (\hat{F}_W^{-1} \frac{\partial \hat{\mathbf{T}}_W}{\partial t} \cdot \hat{\nabla}) \hat{\mathbf{v}} \quad (3.7)$$

$$\mathbf{v} \cdot \nabla \mathbf{v} = \nabla \mathbf{v} \mathbf{v} = \hat{\nabla} \hat{\mathbf{v}} \hat{F}_W^{-1} \hat{\mathbf{v}} = (\hat{F}_W^{-1} \hat{\mathbf{v}} \cdot \hat{\nabla}) \hat{\mathbf{v}} \quad (3.8)$$

$$\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} = \frac{\partial \hat{\mathbf{v}}}{\partial t}(x, t) - (\hat{F}_W^{-1} \frac{\partial \hat{\mathbf{T}}_W}{\partial t} \cdot \hat{\nabla}) \hat{\mathbf{v}} + (\hat{F}_W^{-1} \hat{\mathbf{v}} \cdot \hat{\nabla}) \hat{\mathbf{v}} \quad (3.9)$$

$$= \frac{\partial \hat{\mathbf{v}}}{\partial t} + (\hat{F}_W^{-1} (\hat{\mathbf{v}} - \frac{\partial \hat{\mathbf{T}}_W}{\partial t}) \cdot \hat{\nabla}) \hat{\mathbf{v}} \quad (3.10)$$

The transformation of temporal derivatives, introduces an additional convection term  $(\hat{F}_W^{-1} \frac{\partial \hat{\mathbf{T}}_W}{\partial t} \cdot \hat{\nabla}) \hat{\mathbf{v}}$ , which is accounts for the movement of the domain  $\hat{W}$ .

Moving on to the right hand side of Equation 1.1, we will consider the transformation of the divergence of stress onto the reference domain  $\hat{W}$ . By [?] we have the following relation,

$$\nabla \cdot \sigma = \nabla \cdot (\hat{\mathbf{J}}_W \hat{\sigma} \hat{F}_W^{-T}) \quad (3.11)$$

Where  $\hat{\mathbf{J}}_W \hat{\sigma} \hat{F}_W^{-T}$  is the *first Piola Kirchhoff* stress tensor, relating forces from a Eulerian description of motion to the reference domain  $\hat{W}$ . Assuming a Newtonian fluid, the *Cauchy stress tensor* takes the form  $\sigma = -pI + \mu_f(\nabla \mathbf{v} + (\nabla \mathbf{v})^T)$ . Since  $\sigma \neq \hat{\sigma}$  in  $\hat{W}$ , the spatial derivatives must be transformed, by using Lemma 2.2

$$\begin{aligned} \sigma &= -pI + \mu_f(\nabla \mathbf{v} + (\nabla \mathbf{v})^T) \\ \hat{\sigma} &= -\hat{p}I + \mu_f(\hat{\nabla} \hat{\mathbf{v}} \hat{F}_W^{-1} + \hat{F}_W^{-T} \hat{\nabla} \hat{\mathbf{v}}^T) \end{aligned}$$

For the conservation of continuum we apply the *Piola Transformation* [?] such that

$$\nabla \cdot \mathbf{v} = \nabla \cdot (\hat{\mathbf{J}} \hat{F}_W^{-1} \hat{\mathbf{v}}) \quad (3.12)$$

As the central concepts for transforming the fluid problem on an arbitrary reference domain are introduced, the notation  $\hat{W}$  will no longer be used, instead replaced with the fluid domain  $\hat{\Omega}_f$ , inheriting all previous concepts presented in reference with  $\hat{W}$ . Let  $\hat{T}_f : \hat{\Omega}_f \rightarrow \Omega_f(t)$  be an invertible mapping, with the scalar  $\hat{f}(\hat{x}_f, t) = f(x, t)$  and  $\hat{\mathbf{v}}_f(\hat{x}_f, t) = \mathbf{v}_f(x, t)$  counterparts. Further let  $\hat{F}_f$  be the deformation gradient and  $\hat{J}_w$  its determinant.

**Equation 3.1.1.** ALE fluid problem

Let  $\hat{\mathbf{v}}_f$  be the fluid velocity,  $\rho_f$  the fluid density, and  $\nu_f$  the fluid viscosity.

$$\hat{J}_f \frac{\partial \hat{\mathbf{v}}}{\partial t} + \hat{J}_f (\hat{F}_f^{-1} (\hat{\mathbf{v}} - \frac{\partial \hat{T}_w}{\partial t}) \cdot \hat{\nabla}) \hat{\mathbf{v}} = \nabla \cdot (\hat{J}_w \hat{\sigma} \hat{F}_w^{-T}) + \rho_f \hat{J} \mathbf{f}_f \quad \text{in } \hat{\Omega}_f \quad (3.13)$$

$$\nabla \cdot (\hat{J} \hat{F}_w^{-1} \hat{v}) = 0 \quad \text{in } \hat{\Omega}_f \quad (3.14)$$

where  $\mathbf{f}_s$  represents any exterior body force.

Due to the arbitrary nature of the reference system  $\hat{W}$ , the physical velocity  $\hat{\mathbf{v}}$  and the velocity of arbitrary domain  $\frac{\partial \hat{W}_w}{\partial t}$  doesn't necessary coincide, as it deals with three different reference domains [?]. The Lagrangian particle tracking  $x \in \hat{\Omega}_f$ , the Eulerian tracking  $x \in \Omega_f$ , and the arbitrary tracking of the reference domain  $x \in \hat{W}$  [?]. This concept can be further clarified by the introduction of *material* and *spatial* points.

### 3.1.2 ALE formulation of the solid problem

With the introduced mapping identities we have the necessary tools to derive a full fluid-structure interaction problem defined of a fixed domain. Since the structure already is defined in its natural Lagrangian coordinate system, no further derivations are needed for defining the total problem.

**Equation 3.1.2.** ALE solid problem

$$\rho_s \frac{\partial \hat{\mathbf{v}}_s}{\partial t} = \nabla \cdot \mathbf{FS} + \rho_s \mathbf{f}_s \quad \text{in } \Omega_s \quad (3.15)$$

$$(3.16)$$

### 3.1.3 Fluid mesh movement

Let the total domain deformation  $\hat{T}(\hat{x}, t)$  be divided into the solid  $\hat{T}_s : \hat{\Omega}_s \rightarrow \Omega_s$ , and fluid deformation  $\hat{T}_f : \hat{\Omega}_f \rightarrow \Omega_f$ . The physical motivated solid domain deformation, defined as  $\hat{T}_s : \hat{x}_s + \hat{\mathbf{u}}_s$  where  $\hat{\mathbf{u}}_s$  is the structure deformation, is a consistent mapping from the *reference configuration* to the *current configuration* of the solid domain. As pointed out in section 2.2.2, the deformation of the fluid domain doesn't inherit any physical relation between the two configurations. Despite this fact, one still introduce a fluid deformation variable  $\hat{\mathbf{u}}_f$ , letting the fluid domain transformation be given by

$$\hat{T}_f(\hat{x}, t) = \hat{\mathbf{x}} + \hat{\mathbf{u}}_f(\hat{x}, t)$$

The construction of  $\hat{T}_f(\hat{\mathbf{x}}, t)$  remains arbitrary, however the interface shared by both the fluid and solid domain, require an accurate transformation of the interface points by  $\hat{T}_f$  [?],

$$\hat{T}_f(\hat{\mathbf{x}}, t) = \hat{T}_s(\hat{\mathbf{x}}, t) \quad \leftrightarrow \quad \hat{\mathbf{x}} + \hat{\mathbf{u}}_f(\hat{x}, t) = \hat{\mathbf{x}} + \hat{\mathbf{u}}_s(\hat{x}, t)$$

Therefore the fluid deformation  $\hat{\mathbf{u}}_f$  must have a continuous relation to the structure deformation  $\hat{\mathbf{u}}_s$ , enforced by  $\hat{\mathbf{u}}_f = \hat{\mathbf{u}}_s$  on the interface. For the non-moving boundaries in the fluid domain, tangential deformation are allowed, however normal deformations in relation to the boundaries are not allowed [?]. The fluid domain deformation  $\hat{\mathbf{u}}_f$  must therefore fulfill the boundary conditions

$$\hat{\mathbf{u}}_f(\hat{x}) = \hat{\mathbf{u}}_s \quad \hat{x} \in \hat{\Omega}_f \cup \hat{\Omega}_s \quad (3.17)$$

$$\hat{\mathbf{u}}_f(\hat{x}) \cdot \hat{\mathbf{n}} = 0 \quad \hat{x} \in \partial\hat{\Omega}_f \neq \hat{\Omega}_f \cup \hat{\Omega}_s \quad (3.18)$$

In accordance with conditions 2.17, 2.18, the fluid transformation  $\hat{T}_f(\hat{\mathbf{x}}, t)$  is constructed such that  $\hat{\mathbf{u}}_f$  is an extension of the solid deformation  $\hat{\mathbf{u}}_s$  into the fluid domain. The extension is constructed by a partial differential equation, called a *mesh motion model*.

### 3.1.4 Mesh motion models

In the ALE framework one of the most limiting factors is the degeneration of the mesh due to large deformations. Even the most advanced mesh motion model reaches a limit when only re-meshing is necessary to avoid mesh entanglement [?]. Consequently, the choice of mesh moving technique is essential to generate a smooth evolution of the fluid mesh. Several mesh models have been proposed throughout the literature, and for an overview the reader is referred to [?], and the reference therein. In this thesis, the 2nd order *Laplacian* and *pseudo-elasticity* mesh model, together with the 4th order biharmonic mesh model will be considered. The 2nd order *Laplacian* and *pseudo-elasticity* mesh model are beneficial in terms of simplicity and computational efficiency, at the cost of the regularity of the fluid cells [?]. Hence, the 2nd order models are only capable of handling moderate fluid mesh deformations. Using geometrical or mesh position dependent parameters, the models can be improved to handle a wider range of deformations, by increasing the stiffness of the cell close to the interface ([?], [?]).

A limitation of the 2nd order mesh models is that by Dirichlet and Neumann boundary conditions, only mesh position or normal mesh spacing can be specified respectively, but not both [?]. This limitation is overcome by 4th order biharmonic mesh model, since two boundary conditions can be specified at each boundary of the fluid domain [?]. The 4th order biharmonic mesh model is superior for handling large fluid mesh deformations, as the model generates a better evolution of the fluid cells. A better regularity of the fluid cells also have the potential of less Newton steps needed for convergence at each time-step [?], discussed in section 5.5. The model is however much more computationally expensive compared to the 2nd order mesh models.

### Mesh motion by a Laplacian model

**Equation 3.1.3.** The Laplace equation model

Let  $\hat{\mathbf{u}}_f$  be the fluid domain deformation,  $\hat{\mathbf{u}}_s$  be the structure domain deformation, and let  $\alpha$  be diffusion parameter raised to the power of some constant  $q$ . The Laplacian mesh model is given by,

$$\begin{aligned} -\hat{\nabla} \cdot (\alpha^q \hat{\nabla} \hat{\mathbf{u}}) &= 0 \quad \hat{\Omega}_f \\ \hat{\mathbf{u}}_f &= \hat{\mathbf{u}}_s \quad \text{on } \Gamma \\ \hat{\mathbf{u}}_f &= 0 \quad \text{on } \partial\hat{\Omega}_f/\Gamma \end{aligned}$$

The choice of diffusion parameter is often problem specific, as selective treatment of the fluid cells may vary from different mesh deformation problems. For small deformations, the diffusion-parameter  $\alpha$  can be set to a small constant [[?], [?]]. To accommodate for larger deformations, a diffusion-parameter dependent of mesh parameters, such as fluid cell volume [?] or the jacobian of the deformation gradient [?] have proven beneficial. In [?], the authors reviewed several options based on the distance to the closest moving boundary. This approach will be used in this thesis, using a diffusion-parameter inversely proportional to the magnitude of the distance  $x$ , to the closest moving boundary,

$$\alpha(x) = \frac{1}{x^q} \quad q = -1$$

### Mesh motion by a Linear elastic model

**Equation 3.1.4.** The linear elastic model

Let  $\hat{\mathbf{u}}_f$  be the fluid domain deformation,  $\hat{\mathbf{u}}_s$  be the structure domain deformation, and let  $\sigma$  be the Cauchy stress tensor.  $q$ . The linear elastic mesh model is given by,

$$\begin{aligned} \nabla \cdot \sigma &= 0 \quad \hat{\Omega}_f \\ \hat{\mathbf{u}}_f &= \hat{\mathbf{u}}_s \quad \text{on } \Gamma \\ \hat{\mathbf{u}}_f &= 0 \quad \text{on } \partial\hat{\Omega}_f/\Gamma \\ \sigma &= \lambda \text{Tr}(\epsilon(\hat{\mathbf{u}}_f))I + 2\mu\epsilon(\hat{\mathbf{u}}_f) \quad \epsilon(u) = \frac{1}{2}(\nabla u + \nabla u^T) \end{aligned}$$

Where  $\lambda, \mu$  are Lamé's constants given by Young's modulus  $E$ , and Poisson's ratio  $\nu$ .

$$\lambda = \frac{\nu E}{(1 + \nu)(1 - 2\nu)} \quad \mu = \frac{E}{2(1 + \nu)}$$

The fluid mesh deformation characteristics are in direct relation which the choice of the material specific parameters, Young's modulus  $E$  and Poisson's ratio  $\mu$ . Young's modulus  $E$  describes the stiffness of the material, while the Poisson's ratio relates how a materials shrinks in the transverse direction, while under extension in the axial direction. However the choice of these parameters have proven not to be

---

consistent, and to be dependent of the given problem. In [?] the author proposed a negative poisson ratio, which makes the model mimic an auxetic material, which becomes thinner in the perpendicular direction when submitted to compression. Another approach is to set  $\nu \in [0, 0.5)$  and let  $E$  be inversely proportional to the cell volume [?], or inverse of the distance of an interior node to the nearest deforming boundary surface [?]. In this thesis, the latter is chosen merely for the purpose of code reuse from the Laplace mesh model. The Young's modulus  $E$  and Poisson's ratio  $\mu$  is then defined as,

$$\nu = 0.1 \quad E(x) = \frac{1}{x^q} \quad q = -1$$

### Mesh motion by a Biharmonic model

**Equation 3.1.5.** The bi-harmonic mesh model

Let  $\hat{\mathbf{u}}_f$  be the fluid domain deformation,  $\hat{\mathbf{u}}_s$  be the structure domain deformation. The biharmonic mesh model is given by,

$$\hat{\nabla}^4 \hat{\mathbf{u}}_f = 0 \quad \text{on } \hat{\Omega}_f$$

By introducing a second variable on the form  $\hat{\mathbf{w}} = -\hat{\nabla}^2 \hat{\mathbf{u}}$ , we get the following system defined by

$$\begin{aligned} \hat{\mathbf{w}} &= -\hat{\nabla}^2 \hat{\mathbf{u}} \\ -\hat{\nabla} \hat{\mathbf{w}} &= 0 \end{aligned}$$

In combination with [?], two types of boundary conditions are proposed. Let  $\hat{\mathbf{u}}_f$  be decomposed by the components  $\hat{\mathbf{u}}_f = (\hat{\mathbf{u}}_f^{(1)}, \hat{\mathbf{u}}_f^{(2)})$ . Then we have

$$\begin{aligned} \textbf{Type 1} \quad \hat{\mathbf{u}}_f^{(k)} &= \frac{\partial \hat{\mathbf{u}}_f^{(k)}}{\partial n} = 0 \quad \partial \hat{\Omega}_f / \Gamma \quad \text{for } k = 1, 2 \\ \textbf{Type 2} \quad \hat{\mathbf{u}}_f^{(1)} &= \frac{\partial \hat{\mathbf{u}}_f^{(1)}}{\partial n} = 0 \quad \text{and} \quad \hat{\mathbf{w}}_f^{(1)} = \frac{\partial \hat{\mathbf{w}}_f^{(1)}}{\partial n} = 0 \quad \text{on } \hat{\Omega}_f^{in} \cup \hat{\Omega}_f^{out} \\ \hat{\mathbf{u}}_f^{(2)} &= \frac{\partial \hat{\mathbf{u}}_f^{(2)}}{\partial n} = 0 \quad \text{and} \quad \hat{\mathbf{w}}_f^{(2)} = \frac{\partial \hat{\mathbf{w}}_f^{(2)}}{\partial n} = 0 \quad \text{on } \hat{\Omega}_f^{wall} \end{aligned}$$

The first type of boundary condition the model can interpreted as the bending of a thin plate, clamped along its boundaries. In addition to prescribed mesh position as the Laplacian and linear-elastic model, an additional constraint to the mesh spacing is prescribed at the fluid domain boundary. The form of this problem has been known since 1811, and its derivation has been connected with names like French scientists Lagrange, Sophie Germain, Navier and Poisson [?].

The second type of boundary condition is advantageous when the *reference domain*  $\hat{\Omega}_f$  is rectangular, constraining mesh motion only in the perpendicular direction of the fluid boundary. This constrain allows mesh movement in the tangential direction of the domain boundary, further reducing distortion of the fluid cells [?].

## 3.2 Discretization of the FSI problem

In this thesis, the finite element method will be used to discretize the coupled fluid-structure interaction problem. It is beyond of scope of this thesis, to thorough dive into the analysis of the finite element method regarding fluid-structure interaction problems. Only the basics of the method, which is necessary in order to define a foundation for problem solving will be introduced.

### 3.2.1 Finite Element method

Let the domain  $\Omega(t) \subset \mathbb{R}^d$  ( $d = 1, 2, 3$ ) be a time dependent domain discretized by a finite number of d-dimensional simplexes. Each simplex is denoted as a finite element, and the union of these elements forms a mesh. Further, let the domain be divided by two time dependent subdomains  $\Omega_f$  and  $\Omega_s$ , with the interface  $\Gamma = \partial\Omega_f \cap \partial\Omega_s$ . The initial configuration  $\Omega(t), t = 0$  is defined as  $\hat{\Omega}$ , defined in the same manner as the time-dependent domain.  $\hat{\Omega}$  is known as the *reference configuration*, and hat symbol will refer any property or variable to this domain unless specified. The outer boundary is set by  $\partial\hat{\Omega}$ , with  $\partial\hat{\Omega}^D$  and  $\partial\hat{\Omega}^N$  as the Dirichlet and Neumann boundaries respectively.

The family of Lagrangian finite elements are chosen, with the function space notation,

$$\hat{V}_{\Omega} := H^1(\Omega) \quad \hat{V}_{\Omega}^0 := H_0^1(\Omega)$$

where  $H^n$  is the Hilbert space of degree n.

Let Problem 2.1 denote the strong formulation. By the introduction of appropriate trial and test spaces of our variables of interest, the weak formulation can be deduced by multiplying the strong form with a test function and taking integration by parts over the domain. This reduces the differential equation of interest down to a system of linear equations. (skriv bedre..)

The velocity variable is continuous through the solid and fluid domain

$$\begin{aligned} \hat{V}_{\Omega, \hat{\mathbf{v}}} &:= \hat{\mathbf{v}} \in H_0^1(\Omega), \quad \hat{\mathbf{v}}_f = \hat{\mathbf{v}}_s \text{ on } \hat{\Gamma}_i \\ \hat{V}_{\Omega, \hat{\boldsymbol{\psi}}} &:= \hat{\boldsymbol{\psi}}^u \in H_0^1(\Omega), \quad \hat{\mathbf{v}}_f = \hat{\mathbf{v}}_s \text{ on } \hat{\Gamma}_i \end{aligned}$$

For the deformation, and the artificial deformation in the fluid domain let

$$\begin{aligned} \hat{V}_{\Omega, \hat{\mathbf{u}}} &:= \hat{\mathbf{u}} \in H_0^1(\Omega), \quad \hat{\mathbf{u}}_f = \hat{\mathbf{u}}_s \text{ on } \hat{\Gamma}_i \\ \hat{V}_{\Omega, \hat{\boldsymbol{\psi}}} &:= \hat{\boldsymbol{\psi}}^v \in H_0^1(\Omega), \quad \hat{\boldsymbol{\psi}}_f^v = \hat{\boldsymbol{\psi}}_s^v \text{ on } \hat{\Gamma}_i \end{aligned}$$

For simplification of notation the inner product is defined as

$$\int_{\Omega} \hat{\mathbf{v}} \cdot \hat{\boldsymbol{\psi}} \, dx = (\hat{\mathbf{v}}, \hat{\boldsymbol{\psi}})_{\Omega}$$

### 3.2.2 Variational Formulation

With the primaries set, we can finally define the discretization of the monolithic coupled fluid-structure interaction problem. For full transparency, variation formulation of all previous suggested mesh motion models will be shown. For brevity, the Laplace and linear elastic model will be shorted such that

$$\begin{aligned}\hat{\sigma}_{\text{mesh}} &= \alpha \nabla \hat{\mathbf{u}}_f \quad \text{Laplace} \\ \hat{\sigma}_{\text{mesh}} &= \lambda \text{Tr}(\epsilon(\hat{\mathbf{u}}_f))I + 2\mu\epsilon(\hat{\mathbf{u}}_f) \quad \text{Linear Elasticity}\end{aligned}$$

Further, only the biharmonic model for the first type of boundary condition will be introduced as the second boundary condition is on a similar form. By the concepts of the Finite-element method, the weak variation problem yields.

**Problem 3.1.** *Coupled fluid structure interaction problem for Laplace and elastic mesh moving model. Find  $\hat{\mathbf{u}}_s, \hat{\mathbf{u}}_f, \hat{\mathbf{v}}_s, \hat{\mathbf{v}}_f, \hat{p}_f$  such that*

$$\begin{aligned}(\hat{\mathbf{J}}_f \frac{\partial \hat{\mathbf{v}}_f}{\partial t}, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} + (\hat{\mathbf{J}}_f (\hat{F}_f^{-1}(\hat{\mathbf{v}}_f - \frac{\partial \hat{\mathbf{T}}_f}{\partial t}) \cdot \hat{\nabla}) \hat{\mathbf{v}}_f, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} + (\hat{\mathbf{J}}_f \hat{\sigma} \hat{F}_f^{-T} \hat{\mathbf{n}}_f, \hat{\boldsymbol{\psi}}^u)_{\hat{\Gamma}_i} \\ - (\hat{\mathbf{J}}_f \hat{\sigma} \hat{F}_f^{-T}, \hat{\nabla} \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} - (\rho_f \hat{\mathbf{J}}_f \hat{\mathbf{f}}_f, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} = 0 \\ (\rho_s \frac{\partial \hat{\mathbf{v}}_s}{\partial t}, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_s} + (\hat{\mathbf{F}} \hat{\mathbf{S}} \hat{\mathbf{n}}_f, \hat{\boldsymbol{\psi}}^u)_{\hat{\Gamma}_i} - (\hat{\mathbf{F}} \hat{\mathbf{S}}, \nabla \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_s} - (\rho_s \hat{\mathbf{f}}_s, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_s} = 0 \\ (\frac{\partial \hat{\mathbf{v}}_s - \hat{\mathbf{u}}_s}{\partial t}, \hat{\boldsymbol{\psi}}^v)_{\hat{\Omega}_s} = 0 \\ (\nabla \cdot (\hat{\mathbf{J}}_f \hat{F}_f^{-1} \hat{\mathbf{v}}_f), \hat{\boldsymbol{\psi}}^p)_{\hat{\Omega}_f} = 0 \\ (\hat{\sigma}_{\text{mesh}}, \hat{\nabla} \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} = 0\end{aligned}$$

**Problem 3.2.** *Coupled fluid structure interaction problem for biharmonic mesh moving model. Find  $\hat{\mathbf{u}}_s, \hat{\mathbf{u}}_f, \hat{\mathbf{v}}_s, \hat{\mathbf{v}}_f, \hat{p}_f$  such that*

$$\begin{aligned}(\hat{\mathbf{J}}_f \frac{\partial \hat{\mathbf{v}}_f}{\partial t}, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} + (\hat{\mathbf{J}}_f (\hat{F}_f^{-1}(\hat{\mathbf{v}}_f - \frac{\partial \hat{\mathbf{T}}_f}{\partial t}) \cdot \hat{\nabla}) \hat{\mathbf{v}}_f, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} + (\hat{\mathbf{J}}_f \hat{\sigma} \hat{F}_f^{-T} \hat{\mathbf{n}}_f, \hat{\boldsymbol{\psi}}^u)_{\hat{\Gamma}_i} \\ - (\hat{\mathbf{J}}_f \hat{\sigma} \hat{F}_f^{-T}, \hat{\nabla} \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} - (\rho_f \hat{\mathbf{J}}_f \hat{\mathbf{f}}_f, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} = 0 \\ (\rho_s \frac{\partial \hat{\mathbf{v}}_s}{\partial t}, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_s} + (\hat{\mathbf{F}} \hat{\mathbf{S}} \hat{\mathbf{n}}_f, \hat{\boldsymbol{\psi}}^u)_{\hat{\Gamma}_i} - (\hat{\mathbf{F}} \hat{\mathbf{S}}, \nabla \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_s} - (\rho_s \hat{\mathbf{f}}_s, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_s} = 0 \\ (\frac{\partial \hat{\mathbf{v}}_s - \hat{\mathbf{u}}_s}{\partial t}, \hat{\boldsymbol{\psi}}^v)_{\hat{\Omega}_s} = 0 \\ (\nabla \cdot (\hat{\mathbf{J}}_f \hat{F}_f^{-1} \hat{\mathbf{v}}_f), \hat{\boldsymbol{\psi}}^p)_{\hat{\Omega}_f} = 0 \\ (\hat{\nabla} \hat{\mathbf{u}}, \hat{\nabla} \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} - (\hat{\mathbf{w}}, \hat{\nabla} \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} = 0 \\ (\hat{\nabla} \hat{\mathbf{w}}, \hat{\nabla} \hat{\boldsymbol{\psi}}^v)_{\hat{\Omega}_f} = 0\end{aligned}$$

for the first type of boundary conditions introduced.

### Coupling conditions

**Equation 3.2.1.** Interface coupling conditions

$$\begin{aligned} \mathbf{v}_f &= \mathbf{v}_s && \text{kinematic boundary condition} \\ (\hat{\mathbf{J}}_W \hat{\sigma} \hat{F}_W^{-T} \hat{\mathbf{n}}_f, \hat{\psi}^u)_{\hat{\Omega}_f} &= (\hat{\mathbf{F}} \hat{\mathbf{S}} \hat{\mathbf{n}}_s, \hat{\psi}^u)_{\hat{\Omega}_s} && \text{dynamic boundary condition} \end{aligned}$$

By a continuous velocity field on the whole domain, the *kinematic* condition is strongly enforced on the interface  $\hat{\Gamma}_i$ . The *dynamic* boundary condition is weakly imposed by omitting the boundary integral from the variational formulation, becoming an implicit condition for the system [15].

## 3.3 One-step $\theta$ scheme

For both the fluid problem and the structure problem, we will base our implementation of a  $\theta$ -scheme. A  $\theta$ -scheme is favourable, making implementation of classical time-steppings schemes simple. For the structure problem,  $\theta$ -scheme takes the form

$$\begin{aligned} \rho_s \frac{\partial \hat{\mathbf{v}}_s}{\partial t} - \theta \nabla \cdot \hat{\mathbf{F}} \hat{\mathbf{S}} - (1 - \theta) \nabla \cdot \hat{\mathbf{F}} \hat{\mathbf{S}} - \theta \rho_s \hat{\mathbf{f}}_s - (1 - \theta) \rho_s \hat{\mathbf{f}}_s &= 0 \\ \frac{\partial \hat{\mathbf{v}}_s}{\partial t} - \theta \hat{\mathbf{u}}_s - (1 - \theta) \hat{\mathbf{u}}_s &= 0 \end{aligned}$$

For  $\theta \in [0, 1]$  classical time-stepping schemes are obtained such as the first-order forward Euler scheme  $\theta = 0$ , backward-Euler scheme  $\theta = 1$ , and the second-order Crank-Nicholson scheme  $\theta = \frac{1}{2}$ . Studying the fluid problem, it is initially simpler to consider the Navier-Stokes equation in an Eulerian formulation rather the ALE-formulation. Following [?], a general time stepping algorithm for the coupled Navier-Stokes equation can be written as

$$\begin{aligned} \frac{1}{\Delta} (\mathbf{u}^{n+1} - \mathbf{u}^n) + B(\mathbf{u}^*) \mathbf{u}^{n+\alpha} - \nu \nabla^2 \mathbf{u}^{n+\alpha} &= -\nabla p + \mathbf{u}^{n+\alpha} \\ \nabla \cdot \mathbf{u}^{n+\alpha} &= 0 \end{aligned}$$

Here  $\mathbf{u}^{n+\alpha}$  is an "intermediate" velocity defined by,

$$\mathbf{u}^{n+\alpha} = \alpha \mathbf{u}^{n+1} + (1 - \alpha) \mathbf{u}^n \quad \alpha \in [0, 1]$$

while  $\mathbf{u}^*$  is on the form

$$\mathbf{u}^* = \mathbf{u}^{n+\vartheta} = \begin{cases} \vartheta \mathbf{u}^{n+1} + (1 - \vartheta) \mathbf{u}^n & \vartheta \geq 0 \\ \vartheta \mathbf{u}^{n-1} + (1 - \vartheta) \mathbf{u}^n & \vartheta \leq 0 \end{cases}$$



At first glance, defining an additional parameter  $\vartheta$  for the fluid problem seems unnessecary. A general mid-point rule by  $\alpha = \vartheta = \frac{1}{2}$ , a second order scheme in time would easily be achieved. However, in [?] an additional second order scheme is obtained by choosing  $\alpha = \frac{1}{2}$ ,  $\vartheta = -1$ , where  $\mathbf{u}^*$  is approximated with an Adam-Bashforth linear method. Making the initial fluid problem linear while maintaining second order convergence is an important result, which have not yet been investigated thorough in literature of fluid-structure interaction. However, in the monolithic ALE method presented in this thesis, the fluid problem will still remain non-linear due to the ALE-mapping of the convective term, but making the overall problem "more linear" in contrary with a second order crank-nicholson scheme. The idea was initially pursued in this thesis but left aside, as discretization of the fluid convective term was not intuitive.

In light of By letting  $\alpha = \vartheta$ ,  $\alpha, \vartheta \in [0, 1]$  for the fluid problem, and generalising the consepts in an ALE context, we derive the one-stepl  $\theta$  scheme found in [?].

**Problem 3.3.** The one-step  $\theta$  scheme Find  $\hat{\mathbf{u}}_s, \hat{\mathbf{u}}_f, \hat{\mathbf{v}}_s, \hat{\mathbf{v}}_f, \hat{p}_f$  such that

$$\begin{aligned}
 & (\hat{\mathbf{J}}^{n,\theta} \frac{\partial \hat{\mathbf{v}}}{\partial t}, \hat{\psi}^u)_{\hat{\Omega}_f} + \\
 & \theta (\hat{\mathbf{J}} \hat{F}_W^{-1} (\hat{\mathbf{v}} \cdot \hat{\nabla}) \hat{\mathbf{v}}, \hat{\psi}^u)_{\hat{\Omega}_f} + (1 - \theta) (\hat{\mathbf{J}} \hat{F}_W^{-1} (\hat{\mathbf{v}} \cdot \hat{\nabla}) \hat{\mathbf{v}}, \hat{\psi}^u)_{\hat{\Omega}_f} \\
 & - (\hat{\mathbf{J}} \frac{\partial \hat{\mathbf{T}}_W}{\partial t} \cdot \hat{\nabla}) \hat{\mathbf{v}}, \hat{\psi}^u)_{\hat{\Omega}_f} - \theta (\hat{\mathbf{J}}_W \hat{\sigma} \hat{F}_W^{-T}, \hat{\nabla} \hat{\psi}^u)_{\hat{\Omega}_f} - (1 - \theta) (\hat{\mathbf{J}}_W \hat{\sigma} \hat{F}_W^{-T}, \hat{\nabla} \hat{\psi}^u)_{\hat{\Omega}_f} \\
 & - \theta (\rho_f \hat{\mathbf{J}} \mathbf{f}_f, \hat{\psi}^u)_{\hat{\Omega}_f} - (1 - \theta) (\rho_f \hat{\mathbf{J}} \mathbf{f}_f, \hat{\psi}^u)_{\hat{\Omega}_f} = 0 \\
 & (\rho_s \frac{\partial \hat{\mathbf{v}}_s}{\partial t}, \hat{\psi}^u)_{\hat{\Omega}_s} + -\theta (\hat{\mathbf{F}} \hat{\mathbf{S}}, \nabla \hat{\psi}^u)_{\hat{\Omega}_s} + -(1 - \theta) (\hat{\mathbf{F}} \hat{\mathbf{S}}, \nabla \hat{\psi}^u)_{\hat{\Omega}_s} \\
 & - \theta (\rho_s \hat{\mathbf{f}}_s, \hat{\psi}^u)_{\hat{\Omega}_s} - (1 - \theta) (\rho_s \hat{\mathbf{f}}_s, \hat{\psi}^u)_{\hat{\Omega}_s} = 0 \\
 & (\frac{\partial \hat{\mathbf{v}}_s}{\partial t} - \theta \hat{\mathbf{u}}_s - (1 - \theta) \hat{\mathbf{u}}_s, \hat{\psi}^v)_{\hat{\Omega}_s} = 0 \\
 & (\nabla \cdot (\hat{\mathbf{J}} \hat{F}_W^{-1} \hat{\mathbf{v}}), \hat{\psi}^p)_{\hat{\Omega}_f} = 0 \\
 & (\hat{\sigma}_{\text{mesh}}, \hat{\nabla} \hat{\psi}^u)_{\hat{\Omega}_f} = 0
 \end{aligned}$$

Deeper analysis in [?], specify to important properties of the one-step *theta* scheme. Firstly, it is unconditionally stable regardless of time step for the interval  $\theta = [\frac{1}{2}, 1]$ .



# Verification and Validation

Computer simulations are in many engineering applications a cost-efficient way for conducting design and performance optimization of physical problems. However, thrusting blindly numbers generated from a computer code can prove to be naive. It doesn't take a lot of coding experience before one realizes the many things that can brake down and produce unwanted or unexpected results. Therefore, *credability* of computational results are essential, meaning the simulation is worthy of belief or confidence [4]. *Verification and validation* (V&V) is the main approach for assessing and the reliability of computational simulations [11]. A thorough discussion of (V&V) concepts and terminology during the last century can be found in [4]. In this thesis, the definitions provided by the *American Society of Mechanical Engineers guide for Verification and Validation in Computational Solid Mechanics* [9] are followed.

**Definition 4.1.** Verification: The process of determining that a computational model accurately represents the underlying mathematical model and its solution.

**Definition 4.2.** Validation: The process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model.

Simplified *verification* considers if one solves the equations right, while *validation* is checking if one solves the right equations for the given problem [6]. To test a computational code for all possible parameters, conditions and applications are simply too time consuming. Verification and validation are therefore ongoing processes, with no clear boundary of completeness unless additional requirements are specified [6]. The goal of this chapter is to verify our implementations using the method of manufactured solution (MMS), addressing validation in a later chapter.

## 4.1 Verification of Code

Within scientific computing a mathematical model is often the baseline for simulations of a particular problem of interest. For scientists exploring physical phenomena, the mathematical model is often on the form of systems of partial differential equations (PDEs). A computer program therefore must evaluate mathematical identities such a differential operators and functions in order to produce accurate solutions of the governing PDE's. Through verification of code, the ultimate goal

is to ensure a computer program truly represents the mathematical model. To accumulate sufficient evidence that a mathematical model is solved correctly by a computer code, it must excel within predefined criteria. If the acceptance criterion is not satisfied, a coding mistake is suspected. Should the code pass the preset criteria, the code is considered verified. Of the different classes of test found in [6], *Order-of-accuracy* (OAA) is regarded as the most rigorous acceptance criterion for verification [8, 6, 1]. The method tests if the discretization error  $E$  is reduced in accordance with the *formal order of accuracy* expected from the numerical scheme. The formal order of accuracy is defined to be the theoretical rate at which the truncation error of a numerical scheme is expected to reduce. The *observed order of accuracy* is the actual rate produced by the numerical solution. For order of convergence tests, the code is assumed to be verified if the observed discretization error is proportional to the formal order of accuracy. By monitoring the discretization error  $E$  by spatial and temporal refinements, OAA tests assumes the error  $E$  can be expressed as,

$$E = C\Delta t^p + D\Delta x^l$$

where  $C$  and  $D$  are constants,  $\Delta t$  and  $\Delta x$  represents the spatial and temporal resolution, while  $p$  and  $l$  is the convergence rate of the numerical scheme. If  $\Delta x$  is small compared to  $\Delta t$ , the spatial discretization error can be neglected, and we can use that to find  $l$ , which is the observed order of convergence for the temporal discretization error. To calculate the error  $E$ , an exact/reference solution is needed which rarely exist for complex mathematical models. The next subsection presents an efficient method for generating such solutions.

#### 4.1.1 Method of manufactured solution

The basis of a convergence test is how to find an exact/reference solution, in order to compute the discretization error  $E$ . However solutions of PDEs are limited, and often simplifications of the original problem are needed to produce analytically solutions. *The method of manufactured solutions* provides a simple yet robust way of making analytic solutions for PDE's. Let a partial differential equation of interest be on the form

$$\mathbf{L}(\mathbf{u}) = \mathbf{f}$$

Here  $\mathbf{L}$  is a differential operator,  $\mathbf{u}$  is variable the of interest, and  $\mathbf{f}$  is some source term. In the method of manufactured solution one first manufactures a solution  $\mathbf{u}$  for the given problem. In general, the choice of  $\mathbf{u}$  will not satisfy the governing equations, producing a source term  $\mathbf{f}$  after differentiation by  $\mathbf{L}$ . The produced source term will cancel any imbalance formed by the manufactured solution  $\mathbf{u}$  of the original problem. Therefore, the manufactured solution can be constructed without any physical reasoning, proving code verification as a purely a mathematical exercise were our only interest is to verify the solution [5].

If the MMS is not chosen properly the test will not work, therefore some guidelines for rigorous verification have been proposed in [1, 8, 5].

- The manufactured solution (MS), should be composed of smooth analytic functions such as exponential, trigonometric, or polynomials.
- The MS should have sufficient number of derivatives, exercising all terms and derivatives of the PDE's.

To deeply verify the robustness of the method of manufactured solution, a report regarding code verification by MMS for CFD was published by Salari and Knupp [8]. This thorough work applied the method for both compressible and incompressible time-dependent Navier-Stokes equation. To prove its robustness the authors deliberately implemented code errors in a verified Navier-Stokes solver by MMS presented in the report. In total 21 blind testcases were implemented, where different approaches of verification frameworks were tested. Of these, 10 coding mistakes that reduces the observed order-of-accuracy was implemented. Here the method of manufactured solution captured all coding mistakes, except one. This mistake would, accordingly to the co-author, been captured if his guidelines for conducting MMS had been followed.

In general, computing the source term  $f$  can be quite challenging and error prone. Therefore, symbolic computation of the sourceterm is advantageous to overcome mistakes which can easily occur when calculating by hand. For construction of the sourceterm  $f$ , the Unified Form Language (UFL) [?] provided in FEniCS Project will be used. COMPUTE VV HERE

#### 4.1.2 Comment on verification of the fluid-structure interaction solver by MMS

In general the MMS does not need to match any physical processes. However, when considering multiphysics problems, such as FSI, the equations has to meet the mathematical criteria of the interface.

*Let  $\hat{\mathbf{v}}_s, \hat{\mathbf{v}}_f$  be the structure and fluid velocity, and let  $\sigma_s, \sigma_f$  be the Cauchy stress tensor for the structure and fluid respectively. Let  $\mathbf{n}_i$  be the normal vector pointing out of the domain  $i$ . We then have the following interface boundary conditions,*

1. Kinematic boundary condition  $\hat{\mathbf{v}}_s = \hat{\mathbf{v}}_f$ , enforced strongly by a continuous velocity field in the fluid and solid domain.
2. Dynamic boundary condition  $\sigma_s \cdot \mathbf{n}_s = \sigma_f \cdot \mathbf{n}_f$ , enforced weakly by omitting the boundary integrals from the weak formulation in problem.

The choice of a MMS is therefore not trivial, as it must fulfill condition 1 and 2, in addition to the divergence-free condition in the fluid, and avoiding cancellation of the ALE-convective term  $\frac{\partial \hat{T}_f}{\partial t}$ . The construction of a MMS for a monolithic FSI problem is therefore out of the scope of this thesis. The struggle is reflected of

the absence of research, regarding MMS for coupled FSI solvers in the literature. The challenge are often disregarded, such as [10], where the verification process is conducted on the fluid and structure solver separately. Instead, the correctness of the coupling is evaluated by the code validation. The approach clearly ease the process, assuming verification of each codeblock is "sufficient" to declare the code verified. In this thesis, the approach found in [10] was followed, but it must be stressed that solving each problem individually is not true verification, in reference to a monolithic approach where the problems are solved at the same time.

## 4.2 Validation of Code

Through *verification*, one can assure that a scientific code implements a mathematical model correctly. However, correctness is unnecessary if the model fails to serve as an accurate representation of the physical problem of interest. By definition 1.2, *Validation* is the act of demonstrating that a mathematical model is applicable for its intended use with a certain degree of accuracy. That is, a mathematical model is validated if it meets some predefined criteria within a specific context. Validation is therefore not intended to portray the model as an absolute truth, nor the best model available [7]. In computational science, validation is conducted by comparing numerical results against existing experimental data. The design of validation experiments vary by the motivation of the of their creators, where validated experiments for computational science can be divided into three groups[11]: (1) To improve fundamental understanding of a physical process, (2) Discovery or enhancement of mathematical models of well known physical processes, (3) to conclude the reliability and performance of systems. The assessment of comparison between numerical results and experimental data, makes *validation* assess a wide range of issues [11]. Is the experiment relevant, and conducted correctly in accordance with prescribed parameters? What about the measurement uncertainty of reference experimental data? These issues must be addressed in order to raise sufficient confidence that the mathematical model is credible for its intended use.

Validation of FSI is demanding due to the number of building blocks composing the full problem. For *interface-tracking* methods such as the ALE-method, validation is not only related to the physical aspects of the model. Even if the fluid and structure models excel well within predefined criteria, the non-physical nature of mesh moving models have proven to affect the numerical solution [14]. At first glance, this effect is surprising as mesh moving models simply describe the evolution of fluid mesh cells from the moving interface. However, each mesh model distributes the fluid cells differently, which in turn may have an important effect when conducting mathematical operations such as gradients.

The numerical benchmark presented in [3] has been chosen for validation of the *One-step  $\theta$*  scheme from chapter 3. The benchmark has been widely accepted throughout the fluid-structure interaction community as a rigid validation benchmark [14, 15, 12, 2]. This is mainly due to the diversity of tests included, challenging all the main components of a fluid-structure interaction scheme. The benchmark is based on the *von Kármán vortex street* [13], where a cylinder is intentionally placed off center in a channel. In [3], an additional elastic flag is placed behind the cylinder, see Figure 4.1

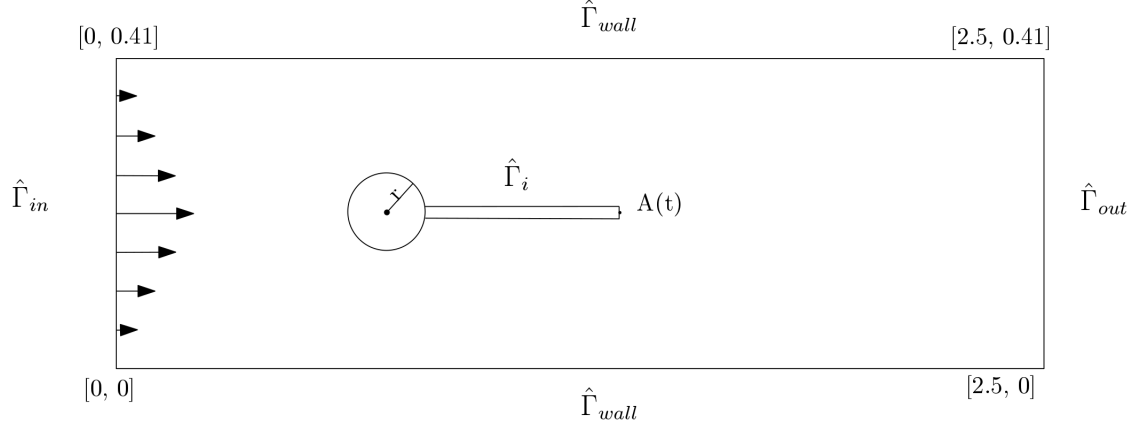


Figure 4.1: Computational domain of the validation benchmark

The benchmark is divided into three main test environments, further divided into three different problems with increasing difficulty. In the first environment the fluid solver is tested for a series of different flow profiles. The second environment considers the structure solver, regarding bending of the elastic flag. And the final environment concerns validation a full fluid-structure interaction problem. Several quantites for comparion are presented in [3] for validation purposes.

- The position  $(x,y)$  of point  $A(t)$  as the elastic flag undergoes deformation.
- Drag and lift forces exerted on of the whole interior geometry in contact with the fluid, consisting of the rigid circle and the elastic beam.

$$(F_D, F_L) = \int_{\Gamma} \sigma \cdot \mathbf{n} dS$$

All environments pose both steady state and periodic solutions. For the steady state solutions, the quantity of interest will be calculated for the last time step. For the periodic solutions, the amplitude and mean values for the time dependent quantity are calculated from the last period of oscillations. The mean value and amplitude is given by,

$$\begin{aligned}\text{mean} &= \frac{1}{2}\text{max} + \text{min} \\ \text{amplitude} &= \frac{1}{2}\text{max} - \text{min}\end{aligned}$$

from the maximum and minimum value of the quantity of interest from the last period. In [3], all steady state solutions seems to be calculated by solving a steady state equation since time-step are only reported for the periodic solutions. In this thesis, all problems in [3] are calculated by time integration. The main motivation is based upon that any given numerical errors regarding time integration will be intercepted at an earlier stage for a simpler problem. Therefore, the choice of time step is chosen such that reasonable accuracy of the reference solution is attained. In the following section, an overview of each environment together with numerical results will be presented. A formal discussion of the results are given at the end of each simulation environment. For each table, the error of the finest spatial and temporal refinement compared to the reference solution is reported in [3], and the followup work.

### 4.2.1 Validation of fluid solver

The first test environment concerns the fluid solver for low Reynold-number regime. Two approaches for the validation are given in [3]. The first approach considers the setup as a fluid-structure interaction problem, by setting the elastic flag close to rigid by manipulation of the structure parameters. In the second approach, the flag is set fully rigid and considered a purely flow problem. By this proposal, no deformation of the fluid domain occurs, reducing the fluid variation formulation to its original form,

$$\begin{aligned}\left(\frac{\partial \hat{\mathbf{v}}_f}{\partial t}, \hat{\boldsymbol{\psi}}^u\right)_{\hat{\Omega}_f} + ((\hat{\mathbf{v}}_f \cdot \hat{\nabla})\hat{\mathbf{v}}_f, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} - (\hat{\sigma}, \hat{\nabla}\hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} - (\rho_f \mathbf{f}_f, \hat{\boldsymbol{\psi}}^u)_{\hat{\Omega}_f} &= 0 \\ (\nabla \cdot \hat{\mathbf{v}}_f, \hat{\boldsymbol{\psi}}^p)_{\hat{\Omega}_f} &= 0\end{aligned}$$

The latter approach is chosen for this thesis, as only the variational formulation for the fluid is tested and removes any influence of the structure and mesh extrapolation discretization.



parameter	CFD 1	CFD 2	CFD 3
$\rho^f [10^3 \frac{kg}{m^3}]$	1	1	1
$\nu^f [10^{-3} \frac{m^2}{s}]$	1	1	1
U	0.2	1	2
Re	20	100	200

Table 4.1: Fluid sub-problem parameters

The validation of the fluid solver is divided into the three sub-cases; CFD1, CFD2, and CFD3, each with different fluid parameters shown in Figure 4.1 While CFD1 and CFD2 yields steady state solutions, CFD3 is a periodic solution. A parabolic velocity profile on the form,

$$v_f(0, y) = 1.5U \frac{(H - y)y}{(\frac{H}{2})^2}$$

is set on the left channel inflow. H is the height of the channel, while the parameter U is set differently to each problem to induce different inlet flow profiles. At the right channel outflow, the pressure is set to  $p = 0$ . No-slip boundary conditions for the fluid are enforced on the channel walls, and on the inner geometry consisting of the circle and the elastic flag. The validation of the fluid solver is based on the evaluation of drag and lift forces on the inner geometry, in comparison with a reference solution. A spatial and temporal convergence study is conducted on all sub-cases The following tables presents the numerical results for each sub-case.

## Results

Table 4.2, 4.3, and 4.4 shows the numerical solution of each sub, CFD-1, CFD-2, and CFD-3. Each sub-problem is evaluated on for four different mesh with increasing resolution. For the numerical solution of CFD-3 in Table 4.4, additional temporal and spatial refinement studies are conducted. Figure 4.1 shows the evaluation of lift and drag for the finest spatial and temporal resolution, while Figure 4.3 shows a visual representation of the fluid flow through the channel.

$\Delta t = 0.1 \quad \theta = 1.0$			
nel	ndof	Drag	Lift
1438	6881	13.60	1.089
2899	13648	14.05	1.126
7501	34657	14.17	1.109
19365	88520	14.20	1.119
Reference		14.29	1.119
Error		0.006 %	0.00 %

Table 4.2: CFD 1 results, lift and drag evaluated at the inner geometry surface for increasing spatial refinement. The error is computed as the relative error from the highest mesh resolution against the reference solution.

$\Delta t = 0.01 \quad \theta = 1.0$			
nel	ndof	Drag	Lift
1438	6881 (P2-P1)	126.0	8.62
2899	13648 (P2-P1)	131.8	10.89
7501	34657 (P2-P1)	135.1	10.48
19365	88520(P2-P1)	135.7	10.55
Reference		136.7	10.53
Error		0.007 %	0.001 %

Table 4.3: CFD 2 results, lift and drag evaluated at the inner geometry surface for increasing spatial refinement. The error is computed as the relative error from the highest mesh resolution against the reference solution.

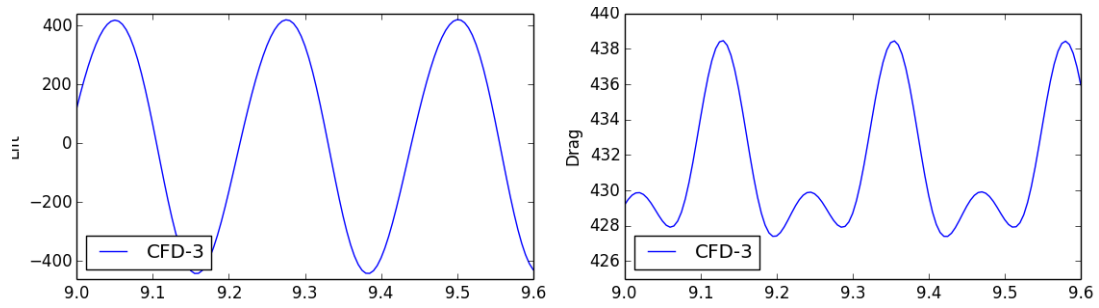


Figure 4.2: CFD-3, lift and drag forces at time  $t = [9, 9.6]$

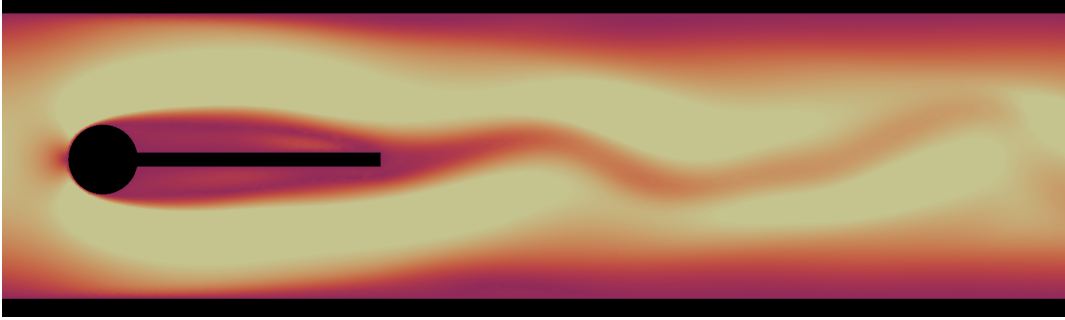


Figure 4.3: CFD-3, flow visualization of velocity time  $t = 9s$

$\Delta t = 0.01 \quad \theta = 0.5$			
nel	ndof	Drag	Lift
1438	6881 (P2-P1)	417.23 +/- 0.0217	-249.21 +/- 0.32
	16474 (P3-P2)	414.86 $\pm$ 5.6282	-7.458 $\pm$ 444.07
2899	13648 (P2-P1)	408.50 $\pm$ 4.3029	-19.731 $\pm$ 373.45
	32853 (P3-P2)	432.86 $\pm$ 5.5025	-9.686 $\pm$ 431.28
7501	34657 (P2-P1)	431.57 $\pm$ 5.2627	-12.497 $\pm$ 429.76
	83955 (P3-P2)	438.20 $\pm$ 5.5994	-11.595 $\pm$ 438.00
19365	88520 (P2-P1)	435.43 $\pm$ 5.4133	-11.545 $\pm$ 438.89
	215219 (P3-P2)	438.80 $\pm$ 5.6290	-11.158 $\pm$ 439.23
Reference		439.95 $\pm$ 5.6183	-11.893 $\pm$ 437.81
Error		0.002 % $\pm$ 0.001 %	0.061 % $\pm$ 0.003%
$\Delta t = 0.005 \quad \theta = 0.5$			
nel	ndof	Drag	Lift
1438	6881 (P2-P1)	417.24 $\pm$ 0.0084	-249.386 $\pm$ 0.1345
1438	16474 (P3-P2)	414.90 $\pm$ 5.7319	-8.467 $\pm$ 443.45
1438	13648 (P2-P1)	408.27 $\pm$ 4.0192	-18.981 $\pm$ 363.84
2899	32853 (P3-P2)	432.90 $\pm$ 5.5333	-11.382 $\pm$ 430.60
1438	34657 (P2-P1)	431.59 $\pm$ 5.2979	-13.644 $\pm$ 429.68
7501	83955 (P3-P2)	438.23 $\pm$ 5.6393	-12.917 $\pm$ 437.78
1438	88520 (P2-P1)	435.46 $\pm$ 5.4579	-13.190 $\pm$ 438.05
19365	215219 (P3-P2)	438.84 $\pm$ 5.6576	-12.786 $\pm$ 438.36
Reference		439.95 $\pm$ 5.6183	-11.893 $\pm$ 437.81
Error		0.002 % $\pm$ 0.006 %	0.075 % $\pm$ 0.001%

Table 4.4: CFD 3 results, lift and drag evaluated at the inner geometry surface. A spatial refinement study is conducted for increasing mesh resolution and two different finite element pairs. The relative error is computed from the solution of the highest mesh resolution, against the reference solution.

## Discussion of results

The numerical solutions of CFD-1 in Figure 4.2 shows convergence against the reference solution. Choosing P2-P1 elements together with a fully implicit scheme  $\theta = 1$ , a relative error of 0.006% for lift, and 0% for drag is attained. For the numerical solution of CFD-2 presented in Figure 4.3, the same observations apply. The second order crank-nicolson scheme  $\theta = 0.5$  was investigated for CFD1 and CFD2, however only improving the results of order  $10^{-6}$  for both lift and drag. For the periodic problem CFD-3, the choice of P2-P1 elements with a fully implicit time-stepping scheme proved insufficient for capturing the expected periodic solution. By crank-nicolson time-stepping scheme  $\theta = 0.5$ , the periodic solution was attained. Since the choice of finite-element pair is not reported in the original work, both P3-P2 and P2-P1 element pairs for fluid and pressure respectively was compared in combination with spatial mesh refinement. From Table 4.3, a relative error  $< 0.08\%$  of the mean and amplitude for lift and drag is attained. The choice P3-P2 element pair is eminent to achieve reasonable results for the first and second mesh regardless of time step. However, the third and fourth mesh resolution shows close resemblance with the reference solution, independent of finite-element pair. On basis of the presented results, the fluid solver is validated in accordance with the proposed benchmark.

### 4.2.2 Validation of solid solver

The validation of the solid solver is conducted on a rectangular domain, representing the elastic structure in Figure 4.1. The structure is submitted to a gravitational for  $\mathbf{g} = (0, g)$ , while being fixed to a fictional wall on the left side of the domain. The validation of the solid solver is based on comparison of the deflection of point  $A(t) = [A_x(t), A_y(t)]$ , conducted on three refined mesh, where the number of finite elements are chosen in close resemblance with the original work in [3]. A simple investigation of different finite-element pairs, suggest that P3-P3 elements were used for making the reference solution. In this study, lower order finite-element pair was included by the motivation of shorter simulation time while retaining solution accuracy. While computational time is not a major concern for the solid solver, the study is important for potentially reducing the computational time for the final validation environment.

parameter	CSM 1	CSM 2	CSM 3
$\rho^s [10^3 \frac{kg}{m^3}]$	1	1	1
$\nu^s$	0.4	0.4	0.4
$\mu^s [10^6]$	0.5	2.0	0.5
$g \frac{m}{s^2}$	2.0	2.0	2.0

Table 4.5: Solid sub-problem parameters

## Results

The numerical results for CSM-1, CSM-2, and CSM-3 are presented in table Table 4.6, 4.7, and 4.8. For the steady state sub-problems CSM-1 and CSM-2, a spatial convergence study is conducted by mesh refinement with three different finite-element pairs . For the periodic CSM-3 problem, an additional temporal study was conducted for two different time steps. in Figure 4.4, a visualization of CSM-3 is provided for three different time steps. Finally, Figure 4.5 shows the displacement vector components, comparing all finite-element pairs for the finest mesh resolution.

$\Delta t = 0.1 \quad \theta = 1.0$			
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]
319	832 P1-P1	-5.278	-56.6
	2936 P2-P2	-7.056	-65.4
	6316 P3-P3	-7.064	-65.5
1365	3140 P1-P1	-6.385	-62.2
	11736 P2-P2	-7.075	-65.5
	25792 P3-P3	-7.083	-65.5
5143	11084 P1-P1	-6.905	-64.7
	42736 P2-P2	-7.083	-65.4
	94960 P3-P3	-7.085	-65.5
Reference		-7.187	-66.1
Error		1.41 %	0.8 %

Table 4.6: CSM-1, deformation components of  $A(t)$  for  $\Delta t = 0.1$  and increasing spatial refinement. The error is computed as the relative error from the highest mesh resolution against the reference solution.

$\Delta t = 0.05 \quad \theta = 1.0$			
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]
319	832 P1-P1	-0.3401	-14.43
	2936 P2-P2	-0.460	-16.78
	6316 P3-P3	-0.461	-16.79
1365	3140 P1-P1	-0.414	-15.93
	11736 P2-P2	-0.461	-16.81
	25792 P3-P3	-0.461	-16.82
5143	11084 P1-P1	-0.449	-16.60
	42736 P2-P2	-0.461	-16.82
	94960 P3-P3	-0.462	-16.82
Reference		-0.469	-16.97
Error		1.49%	0.88 %

Table 4.7: CSM-2, deformation components of  $A(t)$  for  $\Delta t = 0.05$  and increasing spatial refinement. The error is computed as the relative error from the highest mesh resolution against the reference solution.

$\Delta t = 0.01 \quad \theta = 0.5$			
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]
319	832 P1-P1	-10.835 +/- 10.836	-55.197 +/- 56.845
	2936 P2-P2	-14.390 +/- 14.392	-63.303 +/- 65.149
	6316 P3-P3	-14.432 +/- 14.435	-63.397 +/- 65.263
1365	3140 P1-P1	-13.053 +/- 13.054	-60.367 +/- 62.241
	11736 P2-P2	-14.428 +/- 14.432	-63.388 +/- 65.256
	25792 P3-P3	-14.444 +/- 14.446	-63.432 +/- 65.287
5143	11084 P1-P1	-14.082 +/- 14.084	-62.656 +/- 64.495
	42736 P2-P2	-14.444 +/- 14.447	-63.435 +/- 65.288
	94960 P3-P3	-14.449 +/- 14.452	-63.449 +/- 65.296
Reference		-14.305 +/- 14.305	-63.607 +/- 65.160
Error		1% $\pm$ 1%	0.24% $\pm$ 0.24%

$\Delta t = 0.005 \quad \theta = 0.5$			
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]
319	832 P1-P1	-10.846 +/- 10.848	-56.049 +/- 56.053
	2936 P2-P2	-14.390 +/- 14.391	-63.738 +/- 64.703
	6316 P3-P3	-14.429 +/- 14.430	-63.833 +/- 64.810
1365	3140 P1-P1	-13.057 +/- 13.057	-60.813 +/- 61.826
	11736 P2-P2	-14.426 +/- 14.427	-63.827 +/- 64.801
	25792 P3-P3	-14.440 +/- 14.441	-63.854 +/- 64.845
5143	11084 P1-P1	-14.091 +/- 14.091	-63.195 +/- 63.981
	42736 P2-P2	-14.441 +/- 14.441	-63.856 +/- 64.847
	94960 P3-P3	-14.446 +/- 14.446	-63.865 +/- 64.860
Reference		-14.305 +/- 14.305	-63.607 +/- 65.160
Error		1% $\pm$ 1%	0.4% $\pm$ 0.4%

Table 4.8: CSM-3, deformation components of A(t) for increasing temporal and spatial refinement. The error is computed as the relative error from the highest mesh resolution for each choice of time step.

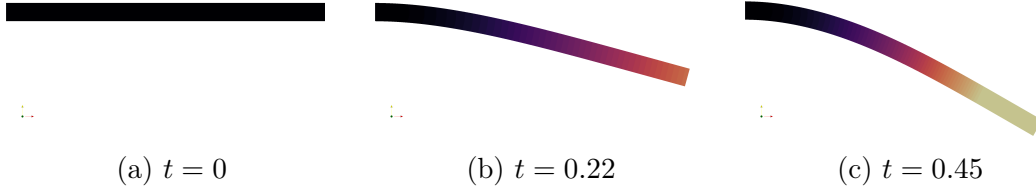


Figure 4.4: CSM-3, visualization of deformation of the elastic flag for three time steps: (a) initial configuration, (b) half way extension, (c) full extension

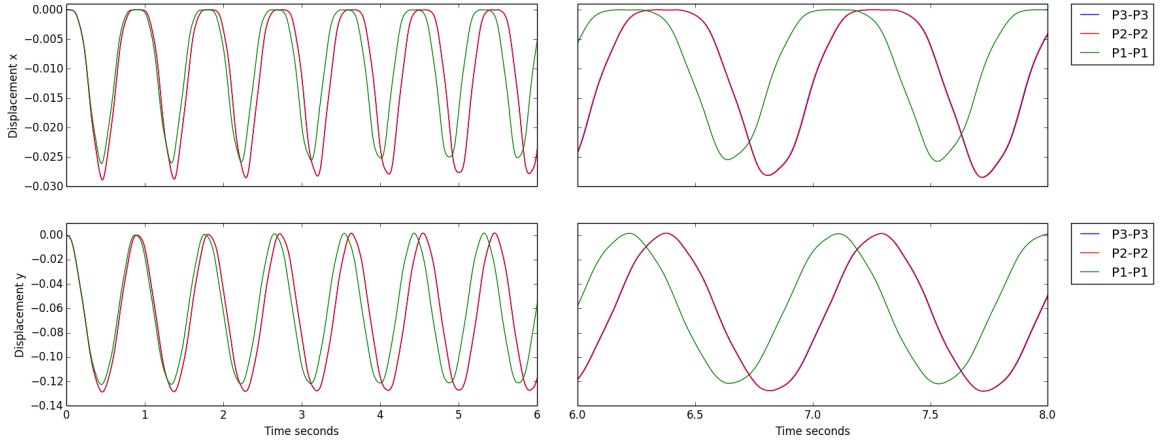


Figure 4.5: CSM-3  $\Delta t = 0.01$ , deformation components of  $A(t)$  with finest mesh resolution, comparing all finite element pairs for time interval  $t \in [0, 6]$  and  $t \in [6, 8]$ .

### Discussion of results

For CSM-1, the relative error of deformation found in Table 4.6, is 1.41% and 0.8% for the x and y coordinate respectively. In Table 4.7, a relative error of 1.49% and 0.88% for the x,y components can be found for CSM-2, proving both steady state problems coincide with the reference solution. In Table 4.8, the numerical solutions CSM-3 for time steps  $\Delta t = 0.01$  and  $\Delta t = 0.005$ , are in close resemblance with the reference solution. The study of lower-grade elements proved successful for all problems, justifying accurate results can be achieved by P2-P2 elements for deformation and velocity, even for course mesh resolution. Comparing all finite-element pairs for CSM-3, visualized in figure 4.5, shows P2-P2 and P3-P3 elements hardly can be distinguished from each other. By the previous mentioned results and observations, the solid solver is assumed validated.

### 4.2.3 Validation of fluid structure interaction solver

The validation of the FSI solver consist of three sub-cases which will be referred to FSI-1, FSI-2 and FSI-3. The FSI-1 environment yields a steady state solution for the system, inducing small deformations to the elastic flag. The FSI-2 and FSI-3 environment results in a periodic solution, where the elastic flag oscillates behind the cylinder. For all sub-cases a parabolic velocity profile on the form,

$$v_f(0, y) = 1.5U \frac{(H - y)y}{(\frac{H}{2})^2}$$

is set on the left channel inflow.  $H$  is the height of the channel, while the parameter  $U$  is set differently to each problem to induce different flow profiles. At the right channel outflow, the pressure is set to  $p = 0$ . No-slip boundary conditions for the fluid are enforced on the channel walls, and on the circle of the inner geometry. The structure deformation and velocity is set to zero on the left side of the flag, where the flag is anchored to the circle. On the fluid-structure interface  $\Gamma$ , we enforce the kinematic and dynamic boundary condition

$$\mathbf{v}_f = \mathbf{v}_s \quad (4.1)$$

$$\sigma_f \cdot \mathbf{n} = \sigma_s \cdot \mathbf{n} \quad (4.2)$$

Apart from the accuracy of the reported values, the main purpose of the validation of the solver is twofold. Firstly, it is of great importance to ensure that the overall coupling of the fluid-structure interaction problem are executed correctly. Second, a good choice of mesh extrapolation model is essential to ensure that mesh entanglement is not present. Based on experience in section, 4.2.1-2, the finite element pair P2-P1 for the fluid solver, and P2-P2 for the solid solver proved successful. Therefore the finite-elements P2-P2-P1 for deformation, velocity, and pressure are chosen for the numerical experiments. Higher order elements will not be examined, mainly due to long computational time, even for optimized solver approaches.



Table 4.9: Benchmark environment

Solid parameters			
parameter	FSI-1	FSI-2	FSI-3
$\rho^s [10^3 \frac{kg}{m^3}]$	1	10	1
$\nu^s$	0.4	0.4	0.4
$\mu^s [10^6 \frac{kg}{ms^2}]$	0.5	0.5	2.0
Fluid parameters			
$\rho^f [10^3 \frac{kg}{m^3}]$	1	1	1
$\nu^f [10^{-3} \frac{m^2}{s}]$	1	1	1
U	0.2	1	2
parameter	FSI-1	FSI-2	FSI-3
Re	20	100	200

## Results

The numerical results for FSI-1, FSI-2, and FSI-3 are shown in Table 4.10-12. For all sub-problems, a spatial convergence study has been conducted on three different mesh with increasing resolution, for all mesh moving models proposed in Chapter 3. For FSI-1 in Table 4.10, an additional option is proposed, omitting a mesh moving model from the monolithic variational form. For FSI-2 and FSI-3, two choices of time steps have been explored with different choice of  $\theta$ .

The relative error of the finest spatial and temporal resolution is provided for the periodic FSI-2 and FSI-3,

## FSI-1

Table 4.10: FSI 1 Results

Laplace					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	0.0226	0.8200	14.061	0.7542
7307	63365	0.0227	0.7760	14.111	0.7517
11556	99810	0.0226	0.8220	14.201	0.7609
Reference		0.0227	0.8209	14.295	0.7638
Error		0.0 %	0.0 %	0.66 %	0.38 %
Linear Elastic					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	0.0226	0.8198	14.061	0.7541
7307	63365	0.0227	0.7762	14.111	0.751
11556	99810	0.0226	0.8222	14.201	0.7609
Reference		0.0227	0.8209	14.295	0.7638
Error		0.0 %	0.0 %	0.66 %	0.38 %
Biharmonic bc1					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	0.0226	0.8200	14.061	0.7541
7307	63365	0.0227	0.7761	14.111	0.7517
11556	99810	0.0227	0.8017	14.205	0.9248
Reference		0.0227	0.8209	14.295	0.7638
Error		0.0 %	0.0 %	0.63 %	21.08 %
Biharmonic bc2					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	0.0226	0.8200	14.061	0.7543
7307	63365	0.0227	0.7761	14.111	0.7518
11556	99810	0.0227	0.8020	14.205	0.9249
Reference		0.0227	0.8209	14.295	0.7638
Error		0.0 %	0.0 %	0.63 %	21.09 %
No extrapolation					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	0.0224	0.9008	14.064	0.7713
7307	63365	0.0226	0.8221	14.117	0.7660
11556	99810	0.0225	0.8787	14.212	0.7837
Reference		0.0227	0.8209	14.295	0.7638
Error		0.0 %	0.0 %	0.58 %	2.61 %

**FSI-2**

Table 4.11: FSI 2 - Comparison of mesh extrapolation models

Laplace $\Delta t = 0.01\theta = 0.51$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-15.27 $\pm$ 13.45	1.34 $\pm$ 82.4	157.00 $\pm$ 14.85	-1.09 $\pm$ 258.47
7307	63365	-14.23 $\pm$ 13.37	1.31 $\pm$ 82.2	159.3 $\pm$ 15.43	0.92 $\pm$ 254.53
11556	99810	-14.96 $\pm$ 13.24	1.28 $\pm$ 81.9	161.07 $\pm$ 17.81	0.02 $\pm$ 256.04
$\Delta t = 0.001\theta = 0.5$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-15.61 $\pm$ 13.21	1.34 $\pm$ 83.6	155.38 $\pm$ 13.98	-3.00 $\pm$ 289.06
7307	63365	-15.31 $\pm$ 13.07	1.02 $\pm$ 82.8	156.81 $\pm$ 14.95	-2.00 $\pm$ 276.24
11556	99810	-15.28 $\pm$ 13.04	1.28 $\pm$ 82.9	158.45 $\pm$ 16.09	-2.53 $\pm$ 276.13
Reference		-14.58 $\pm$ 12.44	1.23 $\pm$ 80.6	208.83 $\pm$ 73.75	0.88 $\pm$ 234.2
Error		0.0% $\pm$ 0.0 %	0.0 % $\pm$ 0.0 %	24.1 % $\pm$ 78.1 %	387.5 % $\pm$ 17.9 %

Biharmonic 1 $\Delta t = 0.01\theta = 0.51$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-15.44 $\pm$ 13.24	-1.38 $\pm$ 82.3	157.67 $\pm$ 15.02	-0.89 $\pm$ 258.87
7307	63365	-15.04 $\pm$ 12.96	0.99 $\pm$ 81.9	159.83 $\pm$ 16.83	0.98 $\pm$ 245.40
11556	99810	-15.29 $\pm$ 13.17	1.29 $\pm$ 82.5	161.69 $\pm$ 18.73	-1.86 $\pm$ 251.30
$\Delta t = 0.001\theta = 0.5$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-15.36 $\pm$ 13.12	1.35 $\pm$ 83.1	155.38 $\pm$ 13.74	-2.55 $\pm$ 285.19
7307	63365	-15.23 $\pm$ 12.97	1.03 $\pm$ 82.4	157.14 $\pm$ 15.18	-8.62 $\pm$ 263.87
11556	99810	-15.27 $\pm$ 12.99	1.31 $\pm$ 82.7	157.72 $\pm$ 15.58	3.34 $\pm$ 258.76
Reference		-14.58 $\pm$ 12.44	1.23 $\pm$ 80.6	208.83 $\pm$ 73.75	0.88 $\pm$ 234.2
Error		0.0% $\pm$ 0.0 %	0.0 % $\pm$ 0.0 %	24.4% $\pm$ 78.8 %	279.5 % $\pm$ 10.4 %

Biharmonic 2 $\Delta t = 0.01\theta = 0.51$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-14.93 $\pm$ 13.22	1.35 $\pm$ 81.5	157.76 $\pm$ 15.04	-0.49 $\pm$ 254.13
7307	63365	-14.67 $\pm$ 13.05	1.00 $\pm$ 80.9	159.59 $\pm$ 16.77	2.22 $\pm$ 248.11
11556	99810	1.58 $\pm$ 12.86	1.23 $\pm$ 81.5	161.85 $\pm$ 18.84	-1.64 $\pm$ 247.04
$\Delta t = 0.001\theta = 0.501$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-15.63 $\pm$ 12.7	1.31 $\pm$ 82.9	155.55 $\pm$ 13.82	-2.45 $\pm$ 281.18
7307	63365	-14.99 $\pm$ 12.81	0.99 $\pm$ 82.14	156.86 $\pm$ 15.05	-1.65 $\pm$ 269.84
11556	99810	-15.26 $\pm$ 12.91	1.27 $\pm$ 81.8	156.86 $\pm$ 15.05	-1.65 $\pm$ 269.84
Reference		-14.58 $\pm$ 12.44	1.23 $\pm$ 80.6	208.83 $\pm$ 73.75	0.88 $\pm$ 234.2
Error		0.0 % $\pm$ 0.0 %	0.0 % $\pm$ 0.0 %	24.8 % $\pm$ 79.5 %	287.5 % $\pm$ 15.2 %

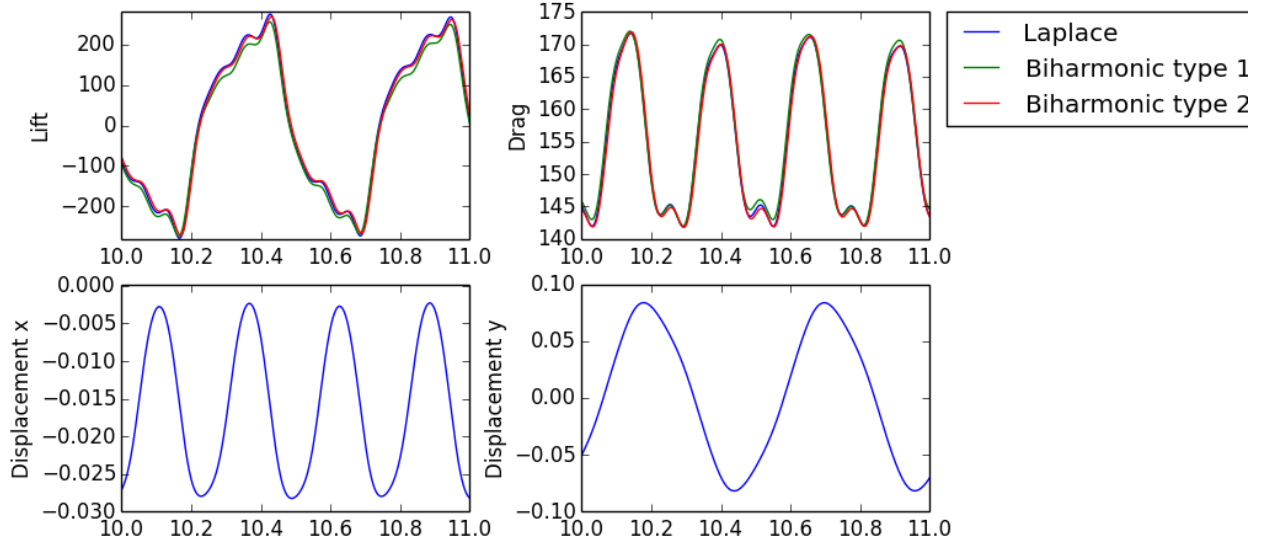


Figure 4.6: FSI-2, visualization of fully developed flow with structure deformation at time  $t = 9s$

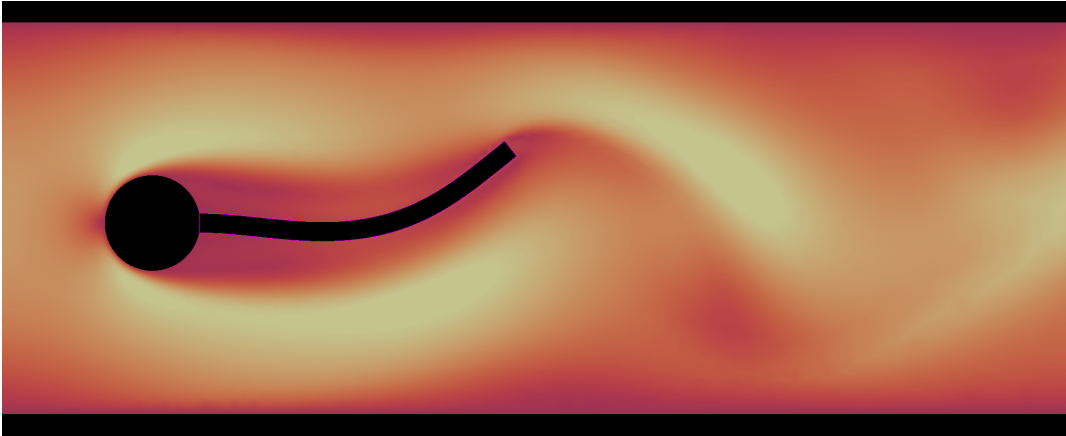


Figure 4.7: FSI-2, visualization of fully developed flow with structure deformation at time  $t = 9s$

**FSI-3**

Table 4.12: FSI 3 - Comparison of mesh extrapolation models

Laplace $\Delta t = 0.01\theta = 0.51$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-2.41 $\pm$ 2.41	1.49 $\pm$ 32.21	449.39 $\pm$ 14.72	0.55 $\pm$ 155.80
7307	63365	-2.32 $\pm$ 2.31	1.32 $\pm$ 31.80	451.76 $\pm$ 16.10	1.04 $\pm$ 151.51
11556	99810	-2.34 $\pm$ 2.34	1.59 $\pm$ 31.91	455.94 $\pm$ 17.34	-0.01 $\pm$ 151.36
$\Delta t = 0.001\theta = 0.5$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-2.91 $\pm$ 2.74	1.28 $\pm$ 35.01	450.90 $\pm$ 18.11	2.28 $\pm$ 161.13
7307	63365	-2.82 $\pm$ 2.66	1.24 $\pm$ 34.69	453.56 $\pm$ 19.80	2.94 $\pm$ 158.67
11556	99810	-2.88 $\pm$ 2.72	1.49 $\pm$ 34.97	458.60 $\pm$ 22.12	2.23 $\pm$ 158.95
Reference		-2.69 $\pm$ 2.56	1.48 $\pm$ 34.38	457.3 $\pm$ 22.66	2.22 $\pm$ 149.78
Error		0.0% $\pm$ 0.0 %	0.0 % $\pm$ 0.0 %	0.28 % $\pm$ 2.38 %	0.45 % $\pm$ 6.12 %

Biharmonic 1 $\Delta t = 0.01\theta = 0.51$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-2.40 $\pm$ 2.38	1.58 $\pm$ 32.07	450.16 $\pm$ 15.11	-20.09 $\pm$ 148.17
7307	63365	-2.26 $\pm$ 2.14	1.70 $\pm$ 31.3	457.37 $\pm$ 15.24	-51.77 $\pm$ 127.28
11556	99810	-2.33 $\pm$ 2.32	1.93 $\pm$ 31.5	456.40 $\pm$ 17.45	0.45 $\pm$ 149.68
$\Delta t = 0.001\theta = 0.5$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-2.18 $\pm$ 2.10	3.52 $\pm$ 2.90	435.19 $\pm$ 9.77	-1.59 $\pm$ 151.45
7307	63365	-2.80 $\pm$ 2.64	1.25 $\pm$ 3.45	454.38 $\pm$ 19.76	17.97 $\pm$ 155.08
11556	99810	-2.84 $\pm$ 2.68	1.50 $\pm$ 3.47	459.12 $\pm$ 22.97	-3.12 $\pm$ 171.22
Reference		-2.69 $\pm$ 2.56	1.48 $\pm$ 34.38	457.3 $\pm$ 22.66	2.22 $\pm$ 149.78
Error		0.0 % $\pm$ 0.0 %	0.0 % $\pm$ 0.0 %	0.40 % $\pm$ 1.37 %	240.5 % $\pm$ 14.3 %

Biharmonic 2 $\Delta t = 0.01\theta = 0.51$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-2.33 $\pm$ 2.33	1.57 $\pm$ 31.6	449.44 $\pm$ 14.82	0.80 $\pm$ 152.03
7307	63365	-2.25 $\pm$ 2.23	1.35 $\pm$ 31.3	452.63 $\pm$ 16.29	17.11 $\pm$ 146.05
11556	99810	-2.25 $\pm$ 2.29	1.59 $\pm$ 31.4	457.89 $\pm$ 17.26	57.83 $\pm$ 141.69
$\Delta t = 0.001\theta = 0.5$					
nel	ndof	ux of A [x 10 <sup>3</sup> ]	uy of A [x 10 <sup>3</sup> ]	Drag	Lift
2474	21249	-2.83 $\pm$ 2.66	1.31 $\pm$ 34.5	450.24 $\pm$ 18.25	2.57 $\pm$ 175.42
7307	63365	-2.77 $\pm$ 2.61	0.98 $\pm$ 34.6	453.53 $\pm$ 20.01	2.60 $\pm$ 159.13
11556	99810	-2.80 $\pm$ 2.65	1.37 $\pm$ 34.7	458.41 $\pm$ 22.23	15.56 $\pm$ 157.78
Reference		-2.69 $\pm$ 2.56	1.48 $\pm$ 34.38	457.3 $\pm$ 22.66	2.22 $\pm$ 149.78
Error		0.0% $\pm$ 0.0 %	0.0% $\pm$ 0.0%	0.24 % $\pm$ 1.90 %	600.9 % $\pm$ 5.34 %

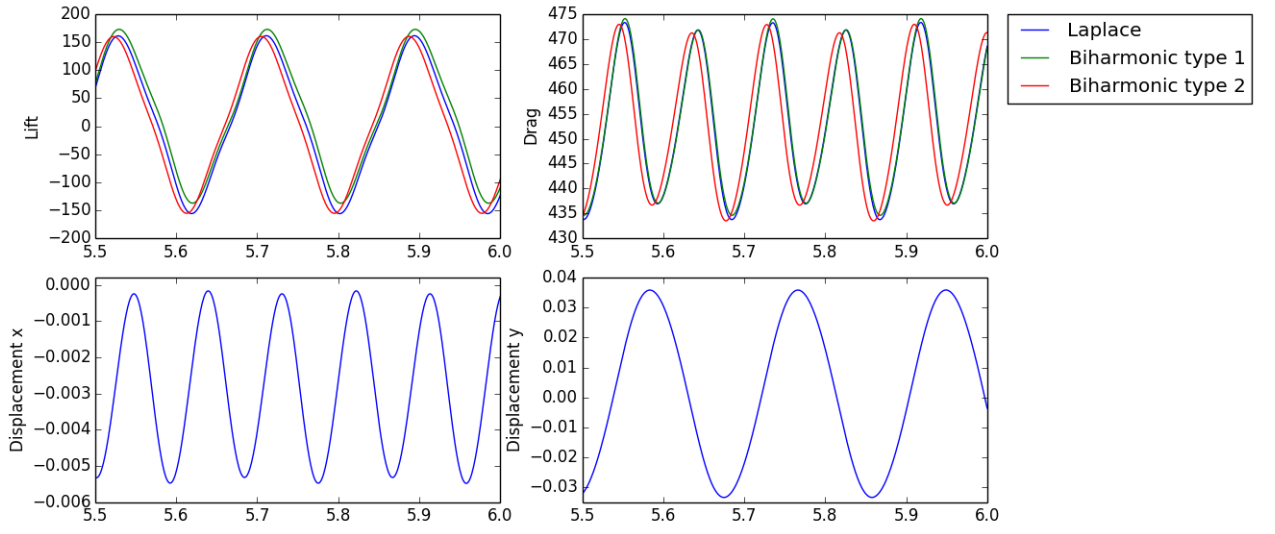


Figure 4.8: Comparison of mesh motion models for FSI-3, in time interval  $t \in [5.5, 6]$

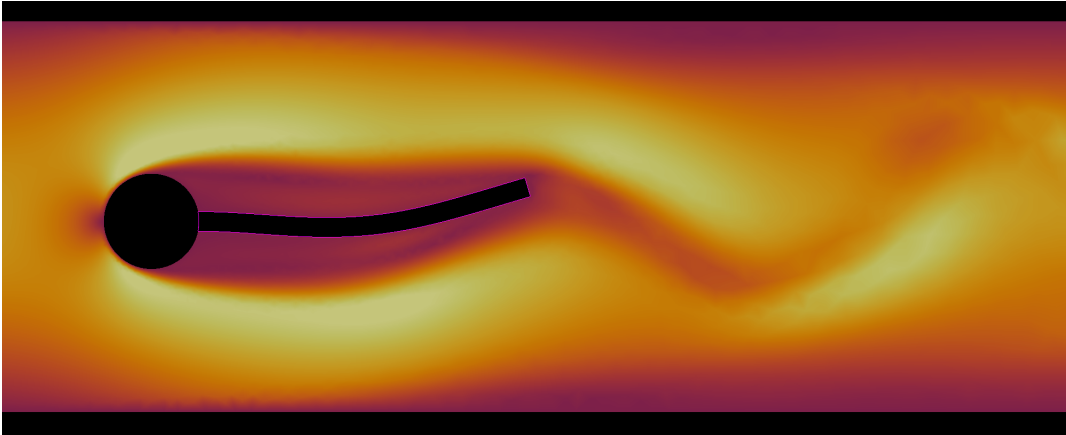


Figure 4.9: FSI-3, visualization of fully developed flow with structure deformation at time  $t = 5.1s$

## Discussion of results

Considering FSI-1, all mesh extrapolation models are of high accuracy compared to the reference solution. However, due to the small deformations of order  $10^{-6}$ , FSI-1 doesn't provide a rigorous test for testing mesh extrapolation models. By omitting mesh extrapolation from the variational formulation, reasonable results are still obtained. Therefore, the FSI-1 validation case can be misleading, in terms of validating the chosen mesh extrapolation model.

The FSI-2 case proved to be one of the most demanding tests, due to the large deformation of the elastic flag, leading to the risk of entangled mesh cells. Therefore a high quality extrapolation of the solid deformation into the fluid is needed. All mesh extrapolation models proved to

The FSI-3 environment does not induce deformation to the extent of the FSI-2. However a critical phase in the transition to the periodic solution was discovered, where the pressure oscillation induces a large deformation to the system.





# Implementation of a Fluid-Structure Interaction solver

This chapter will focus on the implementation of a fluid-structure interaction solver, with emphasis on general implementation issues.

## 5.1 Black-box solvers versus a self-made implementation

A black box is any device whose workings are not understood by or accessible to its user. In terms of computational science, the term is often associated with commercially available code where the user is limited to the control of input parameters, with no intervention of the code itself. Trusting commercial software blindly is risky. Even though the software has been rigorous tested, the lack of full transparency of the implementation ...(full understanding of math). In addition a full understanding of the software also takes time to learn.

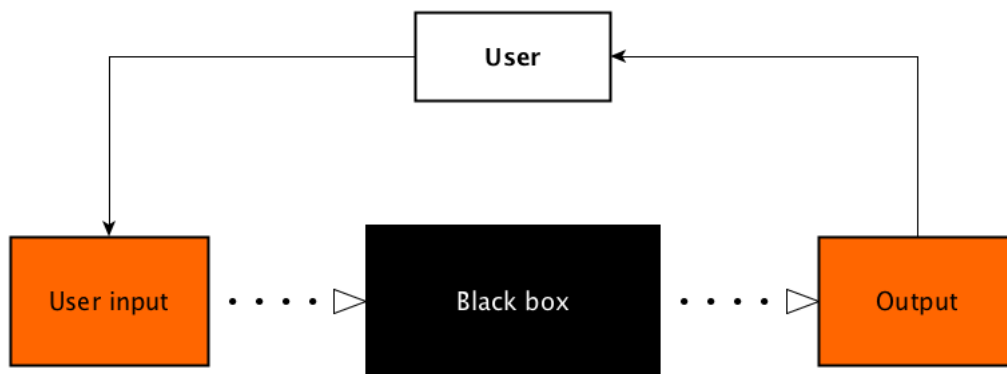


Figure 5.1: Computational domain of the validation benchmark

Several commercial and available solvers for FSI exist, both for monolithic and partitioned approaches. The commercial software ADINA makes solvers based on both approaches available for the user [?]. The open-source OpenFOAM also incorporates both solvers, however enabling more flexibility in combining different

solution strategies ...

The disciplines of CFD and computational structural dynamics (CSD) have traditionally been treated separately. Hence, highly optimized and specialized software has been developed individually, with little or no regard to multiphysics such as fluid-structure interaction. Using a partitioned approach, the MpCCI interface is considered the "de-facto" software, for coupling separate fluid and solid solvers [[?], [?]].

In this thesis a monolithic FSI-solver is developed without the use of commercial available software. The main motivation for this choice is the full understanding of the implementation, and the ability for code intervention along the solution process. The process of implementing a self-made solver for fluid-structure interaction is not straight forward. Due to the overall size of the total system, and the assumption of a non-linear material and large deformations makes the process..

The implementation of the FSI-solver relies solely on the open-source project FEniCS, a computing platform for solving partial differential equations. The next section will present the general concepts for this..

## 5.2 FEniCS

The FEniCS project is an open-source finite element environment for solving partial differential equations (<https://fenicsproject.org/>). Using a combination of high-level Python and C++ interfaces, mathematical models can be implemented compactly and efficiently. FEniCS consists of several sub-modules and we will give a brief overview of the most central components used during implementation and computation.

### 5.2.1 DOLFIN

DOLFIN is the computational C++ backend of the FEniCS project, and the main user interface. It unifies several FEniCS components for implementing of computational mesh, function spaces, functions and finite element assembly.

- UFL (The Unified Form Language) is a domain specific language, used for the discretization of mathematical abstractions of partial differential equations on a finite element form. Its implementation on top of Python, makes it excellent to define problems close to their mathematical notation without the use of more complex features. One uses the term *form* to define any representation of some mathematical problem defined by UFL.
- FFC (The form compiler) compiles the finite elements variation forms given by UFL, generating low-level efficient C++ code
- FIAT the finite element backend, covering a wide range of finite element basis functions used in the discretization of the finite-element forms. It

covers a wide range of finite element basis functions for lines, triangles and tetrahedras.

DOLFIN also incorporate the necessary interfaces to external linear algebra solvers and data structures. Within FEniCS terminology these are called linear algebra backends. PETSc is the default setting in FEniCS, a powerful linear algebra library with a wide range of parallel linear and nonlinear solvers and efficient as matrix and vector operations for applications written in C, C++, Fortran and Python.

A comprehensive introduction FEniCS is out of the scope for this thesis, and for further insight the reader is directed to the introductory demos presented on <https://fenicsproject.org/>.

### 5.2.2 Poission Equation, hello world

When learning programming, it is common to present a "Hello world!" program, presenting the fundamental concepts of the software of interest. For solving PDE's, one of the most basic challenges is solving the Poission's equation. Let  $\Omega$  be the computational domain and let  $\partial\Omega$ , be the boundary of  $\Omega$ . The Poission equation takes the form,

$$\begin{aligned} -\nabla^2 u &= f \quad \in \Omega \\ u &= u_d \quad \in \partial\Omega \end{aligned}$$

where  $u$  is the unknown of interest and  $f$  is some known function.  $\nabla^2$  is the Laplace operator and  $u = u_d$  is a prescribed Dirichlet condition on the boundary. Assuming the reader has some knowledge of the finite-element method, the unknown function  $u$  is approximated by a *trial function*. The problem is then multiplied with a *test function*  $v$ , and then we use integration by-parts over the domain  $\Omega$ .

$$\begin{aligned} - \int_{\Omega} \nabla^2 u v dx &= \int_{\Omega} f dx \\ - \int_{\partial\Omega} \frac{\partial u}{\partial n} v ds + \int_{\Omega} \nabla u \nabla v dx &= \int_{\Omega} f dx \end{aligned}$$

For simplicity, the *test function*  $v = 0$  on the boundary  $\partial\Omega$  due to the prescribed Dirichlet condition which leaves us with the system

$$\int_{\Omega} \nabla u \nabla v dx = \int_{\Omega} f dx$$

With the primaries set, we focus on the implementation of the probelem in FEniCS. The poission problem can then be expressed,

```
1 from fenics import *
2
```

```

3 frame#frame frameCreateframe framemeshframe frameandframe framedefineframe framea
  frame frame1.frameorderframe framelagrangianframe framefunctionframe framespace
4 mesh = UnitSquareMesh(10, 10)
5 V = FunctionSpace(mesh, 'CG', 1)
6
7 frame#frame frameDefineframe framedirichletframe frameboundaryframe framecondition
8 u_bnd = Expression('1 + x[0]*x[0] + 2*x[1]*x[1]', degree=2)
9
10 bcs = DirichletBC(V, u_bnd, "on_boundary")
11
12 Define variational problem
13 u = TrialFunction(V)
14 v = TestFunction(V)
15 f = Constant(-10.0)
16 a = dot(grad(u), grad(v))*dx
17 L = f*v*dx
18
19 frame#frame frameSolveframe frameproblemframe frameandframe framecomputeframe
  framesolution
20 u = Function(V)
21 solve(a == L, u, bc)

```

Algorithm 5.1: possion.py

The possion problem expressed with FEniCS points out two important properties. First, the overall problem is implemented compactly while each code segment for solving the problem remains clear. Second, the abstract notation remains close to the original problem of interest.

## 5.3 implemented code

The overall code structure of the implementation is based on dividing the full code into fragments of Python modules. The main idea is to keep key code segments functionality while maintaining modularity to each code segment.

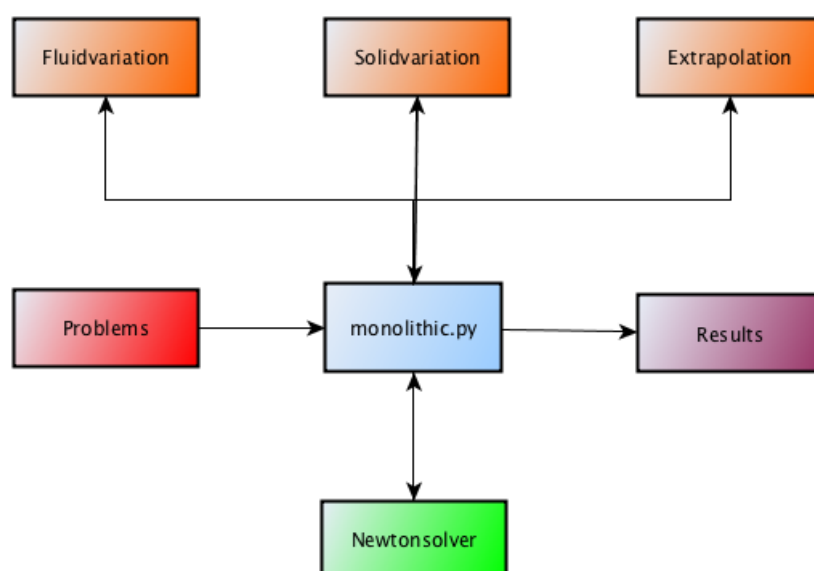


Figure 5.2: Computational domain of the validation benchmark



# Numerical Experiments

## 6.1 Investigation of temporal stability

Preliminary work regarding discretization and numerical analysis of the second order Crank-Nicholson time stepping scheme for fluid structure interaction can be found in [14]. Two main properties of interest have proven to be the stability of long-time simulation, and obtaining the expected physics for the problem of interest. One of the main challenges for constructing time-stepping schemes for ALE-methods, is the additional non-linearity introduced by the domain-velocity term in the fluid problem.

$$\hat{\mathbf{J}}(\hat{F}_f^{-1}(\hat{\mathbf{v}} - \frac{\partial \hat{\Gamma}_f}{\partial t}) \cdot \hat{\nabla})\hat{\mathbf{v}} \quad (6.1)$$

The stability of first and second-order time stepping schemes for ALE-methods have proven to be affected by the ALE-convection term [?, ?], but to what extent remains unclear. Closer inspection of the convection term reveals spatial and temporal differential operators depending non-linearly on one another. These differential operators often appear separated, making discretization of a general time-stepping scheme is not directly intuitive, and often based on the experience of similar equations such as the Navier-Stokes equations.

The Crank-Nicolson scheme can suffer from temporal stability for long-term simulations [?]. Therefore, the authors of [?], investigated temporal stability of the Crank-Nicolson scheme for the validation benchmark found in [3]. The criteria for the numerical experiments was to obtain a stable solution in the time interval [0, 10] minutes, by temporal and spatial refinement studies. The fully monolithic FSI problem discretized with second-order Crank-Nicolson, proved to give general stability problems for long-term simulation for certain time-steps  $k$ . Following the ideas of [?], a second order scheme based on the Crank-Nicolson yields two possibilities.

**Discretization 6.1.** *Crank-Nicolson secant method*

$$\left[ \frac{\hat{\mathbf{J}}(\hat{\mathbf{u}}^n)\hat{\nabla}\hat{\mathbf{v}}^n\hat{\mathbf{F}}_W^{-1}}{2} + \frac{\hat{\mathbf{J}}(\hat{\mathbf{u}}^{n-1})\hat{\nabla}\hat{\mathbf{v}}^{n-1}\hat{\mathbf{F}}_W^{-1}}{2} \right] \frac{\hat{\mathbf{u}}^n - \hat{\mathbf{u}}^{n-1}}{k}$$

**Discretization 6.2.** *Crank-Nicolson midpoint-tangent method*

$$\left[ \frac{\hat{\mathbf{J}}(\hat{\mathbf{u}}_{cn})\hat{\nabla}\hat{\mathbf{v}}_{cn}\hat{\mathbf{F}}_W^{-1}}{2} \right] \frac{\hat{\mathbf{u}}^n - \hat{\mathbf{u}}^{n-1}}{k} \quad \hat{\mathbf{u}}_{cn} = \frac{\hat{\mathbf{u}}^n + \hat{\mathbf{u}}^{n-1}}{2} \quad \hat{\mathbf{v}}_{cn} = \frac{\hat{\mathbf{v}}^n + \hat{\mathbf{v}}^{n-1}}{2}$$

The numerical experiments showed very similar performance for Discretization 1.1 and 1.2, and significant differences of temporal accuracy was not found [?]. Two options to cope with the presented instabilities are the *shifted Crank-Nicolson* [?, ?, ?], and the *frac-step method*. Both of these methods are defined as A-stable time-stepping schemes meaning.. In this thesis the shifted Crank-Nicolson scheme was considered, introducing stability to the overall system by shifting the  $\theta$  parameter slightly to the implicit side. If the shift is dependent of the time-step  $k$  such that  $\frac{1}{2} \leq \theta \leq \frac{1}{2} + k$ , the scheme will be of second order [?].

## Results

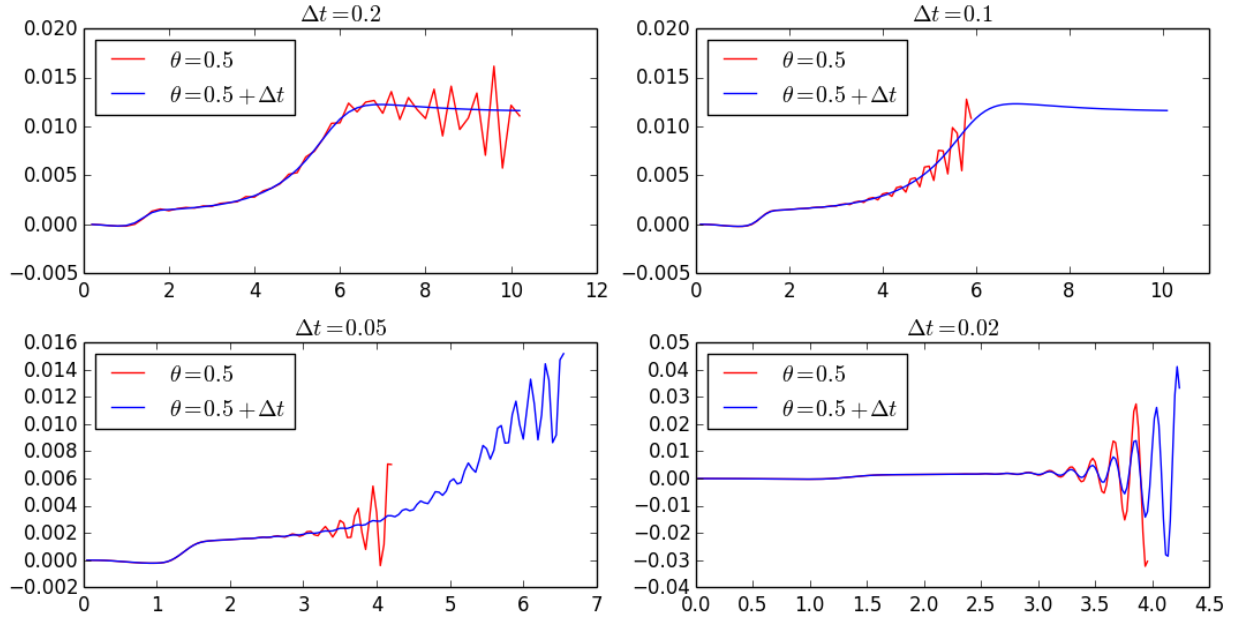


Figure 6.1: Investigation of temporal stability for the FSI3 benchmark in the time interval  $t \in [0, 10]$ , comparing the shifted crank nicolson to the original crank nicolson scheme.



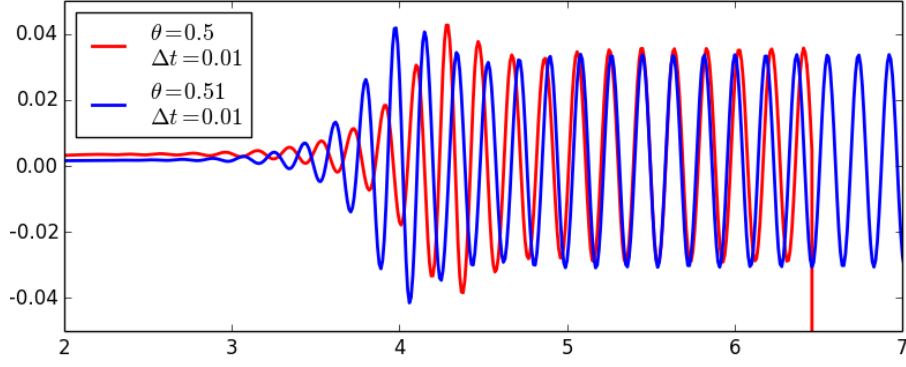


Figure 6.2: Investigation of temporal stability for the FSI3 benchbark in the time interval  $t \in [0, 10]$ , comparing the shifted crank nicolson to the original cranc nicolson scheme.

A numerical investigation of temporal stability in shown in Figure 6.1-2, where the shifted crank-nicolson scheme  $\theta = 0.5 + k$ , is compared the original crank-nicolson  $\theta = 0.5$ . The shifted version clearly show stability properties surpassing the original crank-nicolson scheme, for all numerical experiments. However, for  $\Delta t \in [0.2, 0.1, 0.05, 0.02]$  the scheme clearly lacks the ability to capture the overall physics of the validation problem. Long-time stability and expected physical beavior is obtained for  $\Delta t = 0.01$ . However, numerical experiments showed that for  $\Delta t \leq 0.005$  numerical stability was achieved regardless of both methods. This result is important, reducing the overall computational time needed to achieve reasonable accuracy.

## 6.2 Optimization of Newtonsolver

A *bottleneck* express a phenomena where the total performance of a complete implementation is limited to small code fragments, accounting for the primary consumption of computer resources.

As for many other applications, within computational science one can often assume the consummation of resources follows the *The Pareto principle*. Meaning that for different types of events, roughly 80% of the effects come from 20% of the causes. An analogy to computational sciences it that 80% of the computational demanding operations comes from 20% of the code. In our case, the bottleneck is the newtonsolver. The two main reasons for this is

- **Jacobian assembly**

The construction of the Jacobian matrix for the total residue of the system, is the most time demanding operations within the whole computation.

- **Solver.**

As iterative solvers are limited for the solving of fluid-structure interaction problems, direct solvers was implemented for this thesis. As such, the operation of solving a linear problem at each iteration is computational demanding, leading to less computational efficient operations. Mention order of iterations?

Facing these problems, several attempts was made to speed-up the implementation. The FEniCS project consist of several nonlinear solver backends, were fully user-customization option are available. However one main problem which we met was the fact that FEniCS assembles the matrix of the different variables over the whole mesh, even though the variable is only defined in one to the sub-domains of the system. In our case the pressure is only defined within the fluid domain, and therefore the matrix for the total residual consisted of several zero columns within the structure region. FEniCS provides a solution for such problems, but therefore we were forced to construct our own solver and not make use of the built-in nonlinear solvers.

The main effort of speed-up were explored around the Jacobian assembly. Of the speed-ups methods explored in this thesis, some are *consistent* while others are *nonconsistent*. Consistent methods are methods that always will work, involving smarter approaches regarding the linear system to be solved. The non-consistent method presented involves altering the equation to be solved by some simplification of the system. As these simplifications will alter the expected convergence of the solver, one must take account for additional Newton iterations against cheaper Jacobi assembly. Therefore one also risk breakdown of the solver as the Newton iterations may not converge.

### 6.2.1 Consistent methods

#### Jacobi buffering

By inspection of the Jacobi matrix, some terms of the total residue is linear terms, and remain constant within each time step. By assembling these terms only in the

first Newton iteration will save some assembly time for the additional iterations needed each time step. As consequence the convergence of the Newton method should be unaffected as we do not alter the system.

## 6.2.2 Non-consisten methods

### Reuse of Jacobian

As the assembly of the Jacobian at each iteration is costly, one approach of reusing the Jacobian for the linear system was proposed. In other words, the LU-factorization of the system is reused until the Jacobi is re-assembled. This method greatly reduced the computational time for each time step. By a user defined parameter, the number of iterations before a new assembly of the Jacobian matrix can be controlled.

### Quadrature reduce

The assemble time of the Jacobian greatly depends on the degree of polynomials used in the discretisation of the total residual. Within FEniCS the order of polynomials representing the Jacobian can be adjusted. The use of lower order polynomials reduces assemble time of the matrix at each newton-iteration, however it leads to an inexact Jacobian which may results to additional iterations.

## 6.2.3 Comparison of speedup methods

Table 6.1: Comparison of speedup techniques

Laplace					
Implementation	Naive	Buffering	Reducequad.	Reusejacobi	Combined
Mean time/- timestep	123.1		31.4	61.3	11.1
Speedup %	1.0		74.46%	50.19%	90.97 %
Mean iteration	4.49		10.1	10.2	10.2
Biharmonic Type 1					
Implementation	Naive	Buffering	Reducequad.	Reusejacobi	Combined
Mean time/- timestep	243.3	307.6	51.6	76.7	24.8
Speedup %	1.0	-26%	78.7%	68.4 %	89.7%
Mean iteration	4.1	6.2	4.6	7.1	6.8
Biharmonic Type 2					
Implementation	Naive	Buffering	Reducequad.	Reusejacobi	Combined
Mean time/- timestep			60.5	95.3	20.7
Speedup %	1.0	%	%	%	%
Mean iteration	4.1		6.29	6.9	6.9

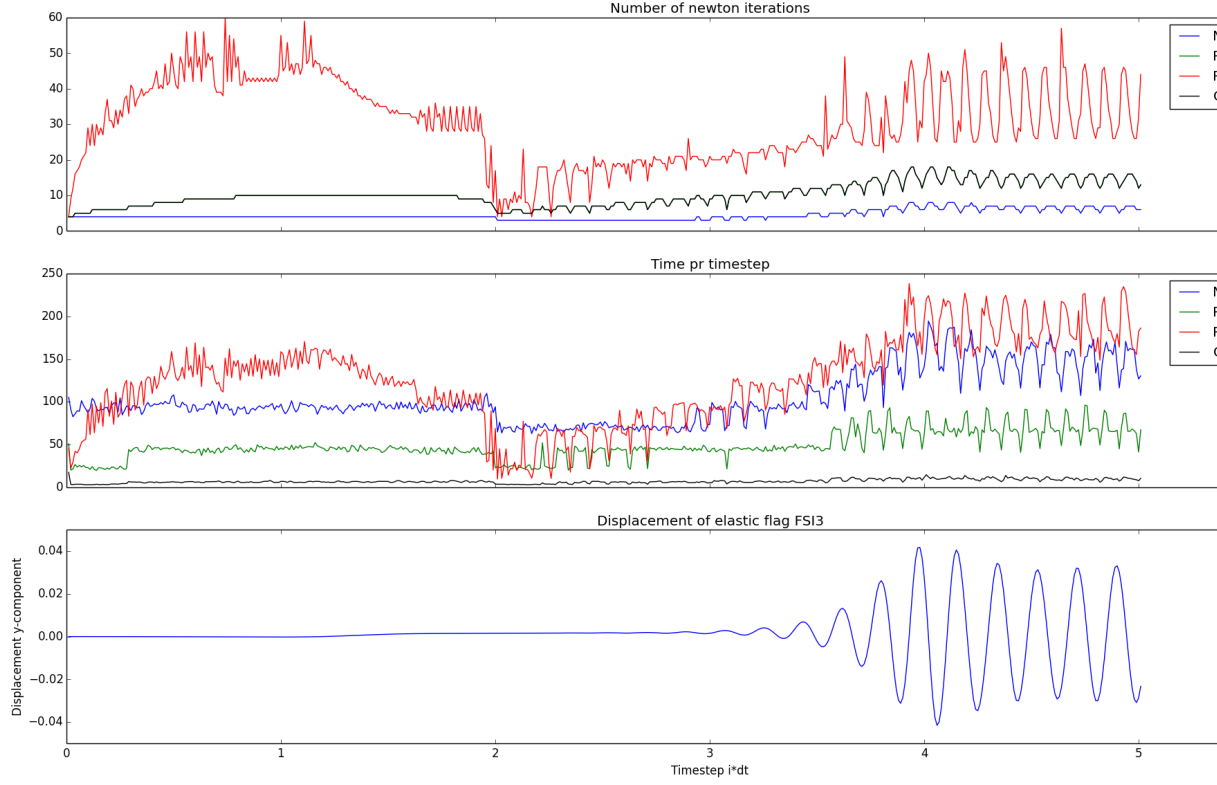


Figure 6.3: Comparison of speed-up techniques for the laplace mesh model

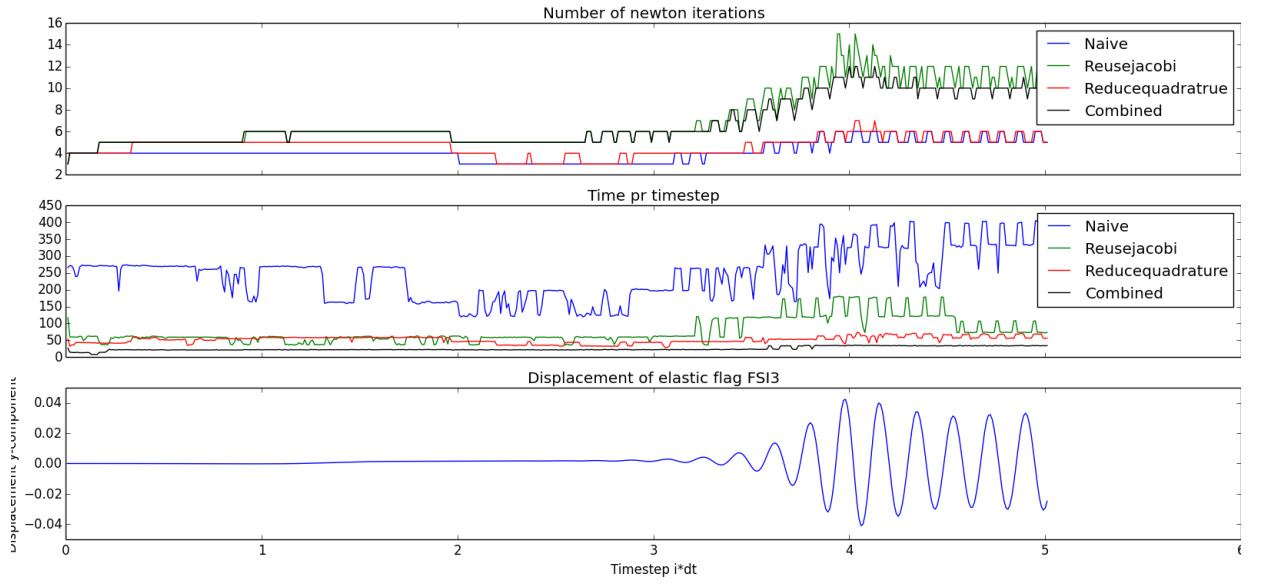


Figure 6.4: Comparison of speed-up techniques for the biharmonic type 1 mesh model

# Conclusion and further research

A monolithic fluid-structure interaction solver have been developed using the arbitrary Lagrangian Eulerian method, suitable for laminar viscous flow and large structure deformation. The code has been verified by order of convergence test, and validated by the fluid-structure interaction benchmark in [3]. A comparison of different mesh moving models have been conducted, with run-time optimizations using a newton solver.

Mesh motion models have proven to effect the physics of the fluid, due to different evolution of fluid mesh cells.

The physics of the fluid have proven to be effected by different mesh motion models,

---

---

# Bibliography

- [1] Stéphane Étienne, D Tremblay, and Dominique Pelletier. Code Verification and the Method of Manufactured Solutions for Fluid-Structure Interaction Problems. *36th AIAA Fluid Dynamics Conference and Exhibit*, (June):1–11, 2006.
- [2] Bernhard Gatzhammer. Efficient and Flexible Partitioned Simulation of Fluid-Structure Interactions. page 261, 2014.
- [3] Jaroslav Hron and Stefan Turek. Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow. *Fluid-Structure Interaction*, 53:371–385, 2006.
- [4] William L. Oberkampf and Christopher J. Roy. *Verification and Validation in Scientific Computing*. Cambridge University Press, Cambridge, 2010.
- [5] Patrick J. Roache. Code Verification by the Method of Manufactured Solutions. *Journal of Fluids Engineering*, 124(1):4, 2002.
- [6] P.J. Roache. *Verification and Validation in Computational Science and Engineering*. Computing in Science Engineering, Hermosa Publishers, 1998, 8-9, 1998.
- [7] Edward J. Rykiel. Testing ecological models: The meaning of validation. *Ecological Modelling*, 90(3):229–244, 1996.
- [8] Kambiz Salari and Patrick Knupp. Code Verification by the Method of Manufactured Solution. Technical report, Sandia National Laboratories, 2000.
- [9] LE Schwer. Guide for verification and validation in computational solid mechanics. *American Society of Mechanical Engineers*, PTC 60(V&V 10):1–15, 2006.
- [10] Jason P Sheldon, Scott T Miller, and Jonathan S Pitt. Methodology for Comparing Coupling Algorithms for Fluid-Structure Interaction Problems. *World Journal of Mechanics*, 4(February):54–70, 2014.
- [11] Ian Sommerville. Verification and Validation. Technical Report February, 2006.
- [12] A V, T Passerini, A Quaini, U Villa, A Veneziani, and S Canic. Numerical Analysis and Scientific Computing P reprint S eria Validation of an open source framework for the simulation of blood flow in rigid and deformable

vessels P reprint # 17 D epartment of Mathematics University of Houston. 2013.

- [13] Frank White. *Viscous fluid flow*. McGraw-Hill, third edit edition.
- [14] T Wick and Thomas Wick. Variational-monolithic ALE fluid-structure interaction: Comparison of computational cost and mesh regularity using different mesh motion techniques.
- [15] Thomas Wick. Solving Monolithic Fluid-Structure Interaction Problems in Arbitrary Lagrangian Eulerian Coordinates with the deal.II Library.