

Politecnico di Milano

SCHOOL OF INDUSTRIAL AND INFORMATION ENGINEERING

Master of Science – Aerospace Engineering



Thesis Title

Supervisor

Title Name SURMANE

Co-Supervisor

Title Name SURNAME

Candidate

Claudio CACCIA – 820091

Academic Year 2019 – 2020

Acknowledgements

Thanks

Abstract

This thesis describes bla bla...

Keywords— one, two, three, four

Sommario

In questa tesi bla bla...

Contents

Acknowledgements	iii
Abstract	v
Sommario	vii
Contents	x
List of Figures	xi
List of Tables	xiii
1 Introduction	1
2 Physical aspects of Fluid-Structure Interaction problems	3
2.1 Description of motion	3
2.1.1 Eulerian perspective	3
2.1.2 Lagrangian perspective	4
2.1.3 ALE method	5
2.2 Domains and interface	6
2.2.1 Fluid domain	6
2.2.2 Solid domain	7
2.2.3 Models with reduced dimensionality: beams	8
2.2.4 Interface and interaction	9
2.3 Classification of FSI problems	10
2.3.1 Dimensional analysis	11
2.3.2 Dimensional analysis in fluid domain	11
2.3.3 Dimensional analysis in solid domain	12
2.3.4 Dimensional analysis of coupled problems	12
3 Computational aspects of Fluid-Structure Interaction problems	15
3.1 Monolithic and Partitioned Approach	15
3.2 Coupling Strategies	16
3.3 Interface Mesh	17
3.4 Stability: Added Mass Effect	18
4 Software Packages used in this work	21
5 MBDyn Adapter and its integration	23
6 Validation Test-cases	25

Contents

7	Conclusions	27
	Conclusions	27
A	First Appendix	31
	Acronyms	37
	Bibliography	40

List of Figures

Figure 2.1	Eulerian perspective	4
Figure 2.2	Lagrangian perspective	4
Figure 2.3	ALE perspective	5
Figure 2.4	ALE mesh	6
Figure 2.5	beam model, taken from [1]	9
Figure 2.6	fluid solid interface	9
Figure 2.7	First Figure	14

List of Tables

Chapter 1

Introduction

Fluid-Structure Interaction (FSI) describes the mutual interaction between a moving or deformable object and a fluid in contact with it, surrounding or internal. It is present in various forms both in nature and in man-made systems: a leaf fluttering in the wind, water flowing underground or blood pumping in an artery are typical examples of fluid-structure interaction in nature. FSI occurs in engineered systems when modeling the behavior of turbomachinery, the flight characteristics of an aircraft, or the interaction of a building with the wind, just to name a few examples.

All the aforementioned problems go under the same category of FSI, even if the nature of the interaction between the solid and fluid is different. Specifically, the intensity of the exchanged quantities and the effect in the fluid and solid domains vary among different problems.

There can be multiple ways to classify FSI problems, based on the flow physics and on the behavior of the body. Incompressible flow assumption is always made for liquid-solid interaction, while both compressible and incompressible flow assumptions are made when a gas interacts with a solid, depending on the flow properties and the kind of simulation. The main application of air-solid interaction considers the determination of aerodynamic forces on structures such as aircraft wings, which is often referred to as *aeroelasticity*. Dynamic aeroelasticity is the topic that normally investigates the interaction between aerodynamic, elastic and inertial forces. Aerodynamic *flutter* (i.e. the dynamic instability of an elastic structure in a fluid flow) is one of the severe consequences of aerodynamic forces. It is responsible for destructive effects in structures and a significant example of FSI problem.

The subject may also be classified considering the behavior of the structure interacting with the fluid: a solid can be assumed rigid or deforming because of the fluid forces. Examples where rigid body assumption may be used include internal combustion engines, turbines, ships and offshore platforms. The rigid body–fluid interaction problem is simpler to some extent, nevertheless the dynamics of rigid body motion requires a solution that reflects the fluid forces. Within the deformable body–fluid interaction, the nature of the deforming body may vary from very simple linear elastic models in small strain to highly complex nonlinear deformations of inelastic materials. Examples of deforming body–fluid interaction include aeroelasticity, biomedical applications and poroelasticity.

The interaction between fluid (incompressible or compressible) and solid (rigid or deformable) can be *strong* or *weak*, depending on how much a change in one domain influences the other. An example of weakly coupled problem is aeroelasticity at high Reynolds number, while incompressible flow often leads to strongly coupled problems. This distinction can lead to different solution strategies, as briefly described below.

Physical models aren't the only way in which FSI problems can be classified. The solution

procedure employed plays a key role in building models and algorithms to solve this kind of problems. The two main approaches are: the *monolithic approach* in which both fluid and solid are treated as one single system and the *partitioned approach* in which fluid and solid are considered as two separated systems coupled only through an interface. This latter approach is often preferred when building new solution procedures as it allows to use solvers that have been already developed, tested and optimized for a specific domain. The solvers only need to be linked to a third component, which takes care of all the interaction aspects.

The partitioned approach can be further classified considering the coupling between the fluid and solid: the solution may be carried out using a *weakly coupled approach*, in which the two solvers advance without synchronization, or a *strongly coupled approach*, in which the solution for all the physics must be synchronized at every time step. Although the weakly coupled approach is used in some aerodynamic applications, it is seldom used in other areas due to instability issues. A strongly coupled approach is generally preferred, even though this leads to more complex coupling procedures at the interface between fluid and solid.

This work describes the implementation and the validation of what is called an *adapter*, that is the "glue-code" needed to interface a solver to a coupling software library, thus adopting a *partitioned approach* to solve FSI problems. The *adapter* presented here connects the software code *MultiBody Dynamics analysis software (MBDyn)* to the multiphysics coupling library *precise Code Interaction Coupling Environment (preCICE)*.

Interfacing MBDyn with preCICE has multiple advantages: on one side it extends the capabilities of MBDyn to be used in FSI simulations by connecting it with a software library which has been already connected to widely used CFD solvers; on the other side, it allows to describe and simulate FSI problems with a suite of lumped, 1D and 2D elements (i.e. rigid bodies, *beams*, *membranes*, *shells*, etc.) decoupling the shape of the object (the interface with the fluid) from its structural properties, which can be described by different models and constitutive laws.

The thesis is structured as follows:

- Section 2 introduces the reader to FSI problems and their complexity, with particular attention to the physical description of the fluid and solid domains and the interface.
- Section 3 focuses on numerical methods, describing how to computationally deal with these kind of problems: details regarding the different coupling approaches are given here.
- Section 4 explains the features of preCICE that the adapter needs to support and gives a short introduction to MBDyn, explaining the main functionalities of interest.
- Section 5 presents the adapter developed in this work, its most important features and how to configure a FSI simulation with it.
- Section 6 describes the successful validation of the adapter, the comparison of the results with some well-known benchmarks and an example of real world application.
- Section 7 summarizes the findings and outcome of this work and gives an outlook to future work on this topic.
- Finally XX appendices give further information on...

Chapter 2

Physical aspects of Fluid-Structure Interaction problems

Dynamic models of solids or fluids aim at describing the evolution of an initial configuration through time. Structural mechanics and fluid dynamics use different perspectives when describing the motion of respectively a solid or a fluid particle. When dealing with FSI problems the two approaches need to be combined in order to obtain a suitable description of the two domains and their interface: this aspect is treated in 2.1.

As outlined in the introduction, the fluid and the solid domain of a FSI problem might be described by means of many different models: some of them are outlined in section 2.2. *Dimensional analysis* and the use of dimensionless numbers is a powerful tool used to classify fluid dynamics problems: some of the principles used there can be applied to FSI problems in order to classify them: this can help define and classify FSI problems, as described in section 2.3.

2.1 Description of motion

In a FSI model, the fluid in motion deforms the solid because of the forces exerted to the structure. The change in the shape of the solid modifies the fluid domain, causing a different flow behavior. For this reason it is necessary to describe formally the kinematics and the dynamics of the whole process. Classical continuum mechanics considers the motion of particles by means of two different perspectives [2]: the *Eulerian description*, briefly described in section 2.1.1, and the *Lagrangian description*, outlined in section 2.1.2. Those two perspectives are typically combined into the *arbitrary Lagrangian-Eulerian (ALE)* method, described in section 2.1.3.

2.1.1 Eulerian perspective

The *Eulerian perspective* observes the change of quantities of interest (e.g. density, velocity, pressure) at spatially fixed locations. In other words: the observer does not vary the point of view during different time steps. Thus, quantities can be expressed as functions of time at fixed locations. This is represented by the following notation:

$$\Theta = \tilde{\Theta}(x, y, z, t) \tag{2.1}$$

where Θ is a quantity of interest and $\tilde{\Theta}$ denotes the same quantity from an Eulerian point of view; (x, y, z) represent a fixed location in the euclidean space.

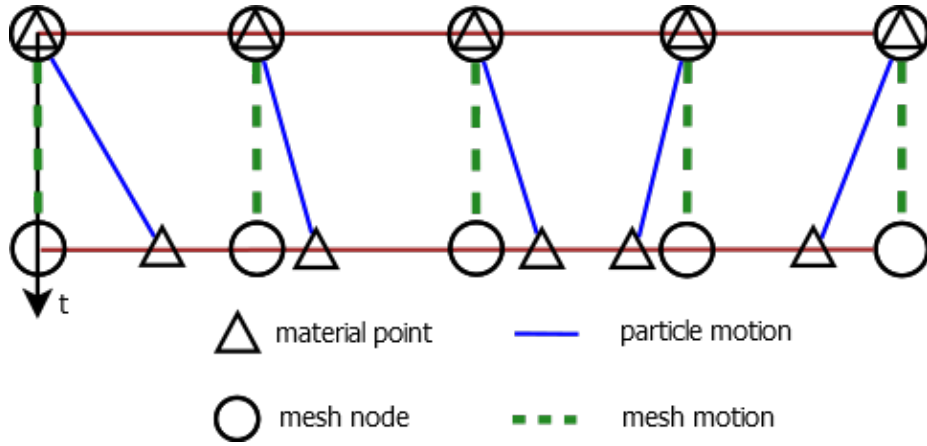


Figure 2.1. Eulerian perspective

A computational mesh can be interpreted as a number of observers distributed across the domain of interest and connected to each other in order to form a grid with nodes. If the particles of the domain move, a purely euclidean mesh does not move and the position other nodes remain fixed at any instance of time [3]. This behavior is represented in Figure 2.1 (adapted from [3]). The mesh is independent of particles movement, resulting in a convenient choice for Computational Fluid Dynamics (CFD) problems, where fluid flows throughout the whole computational domain. Within this approach, proper mesh refinement is crucial for computational accuracy as it defines to what extent small scale movement can be modeled and resolved [4].

2.1.2 Lagrangian perspective

A *lagrangian observer* focuses on a single particles and follows it throughout the motion, as depicted in Figure 2.2. Changes in the quantities of interest are observed at different spatial locations.

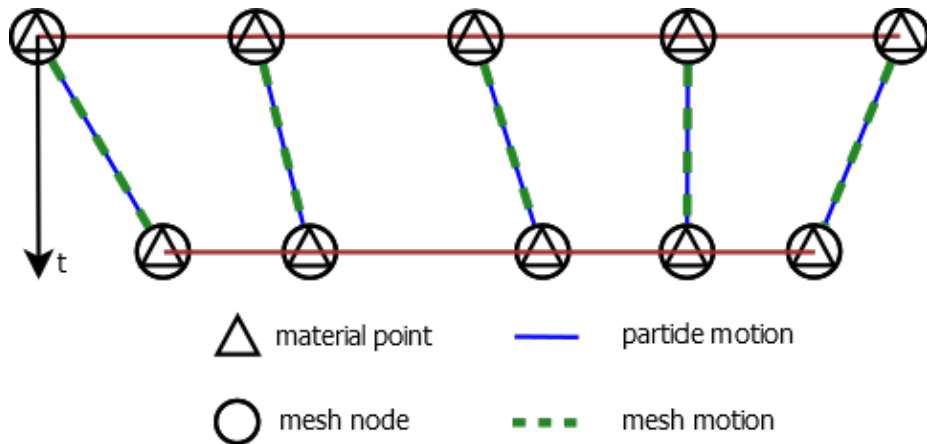


Figure 2.2. Lagrangian perspective

The motion of the particle and the other quantities of interest can be described by reference coordinates (or *material coordinates*) in Euclidean space (X, Y, Z) , uniquely identifying the observed particle at a reference configuration [5]. Usually $t = 0$ is chosen as reference but this

is not mandatory. The Lagrangian observer only registers changes concerning one specific particle as time advances. Thus, quantities of interest can be described as:

$$\Theta = \hat{\Theta}(X, Y, Z, t) \quad (2.2)$$

In contrast to the Eulerian perspective (Equation 2.1), the obtained information is strictly limited to a single material particle (implied by the usage of the capital reference coordinate variables). Information about a fixed point in space is not directly available and no convective fluxes appear in a Lagrangian description.

This perspective is again translated into computational meshes: at a reference instance of time, mesh nodes are attached to material particles. As these move, the mesh nodes move with them causing the mesh to deform. Figure 2.2 describes the situation. The mesh nodes always coincide with their respective particles.

In this situation large-scale and irregular motions and more importantly deformation lead to distortions of the computational mesh, which yields smaller accuracy in simulations requiring to apply techniques to keep the desired accuracy [6].

Lagrangian perspective is the usual method of choice for Computational Solid Mechanics (CSM) simulations.

Eulerian and Lagrangian descriptions are related [7]. A mapping between them can be described by the *motion* function ϕ such that:

$$\vec{x}(t) = \phi(\vec{X}, t) \quad (2.3)$$

Equation 2.3 tells that the Eulerian, spatial position \vec{x} of a particle at time t is the mapping of the particle at its reference configuration \vec{X} : the mapping must be bijective.

2.1.3 ALE method

As outlined above, CSM and CFD problems adopt different perspectives. The ALE approach, a combination of the two points of view, is used for FSI problems. As the name implies, an ALE observer moves arbitrarily with respect to a specific material particle. Figure 2.3 depicts such a situation.

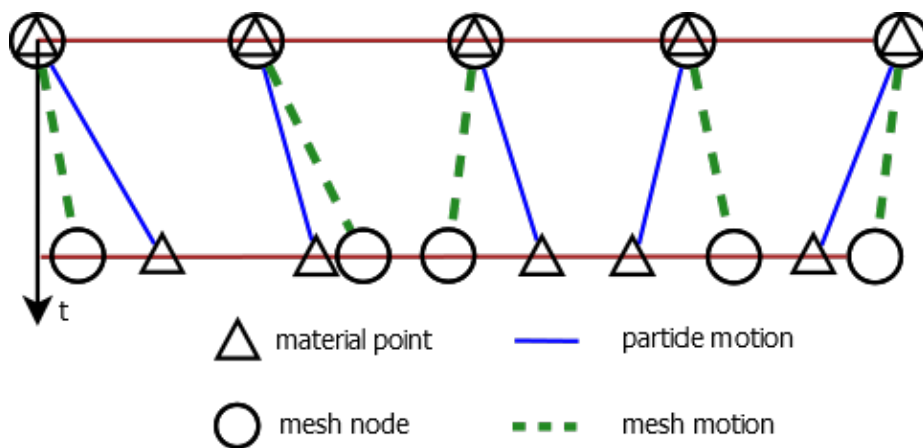
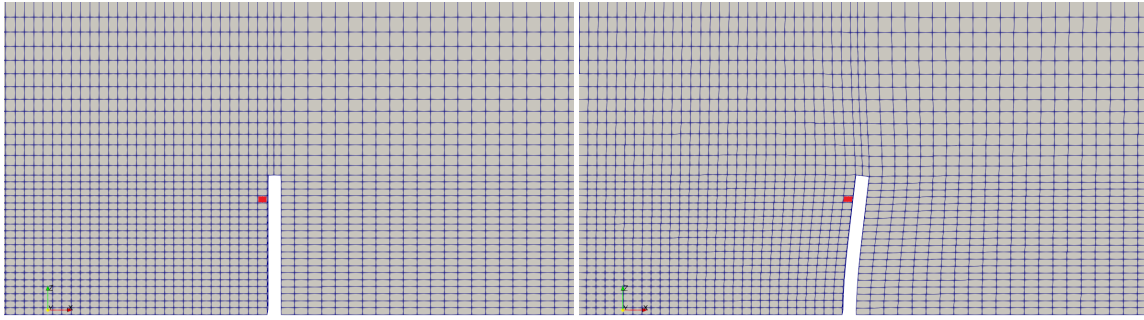


Figure 2.3. ALE perspective

When dealing with computational meshes, an ALE mesh is considered as it can move almost arbitrarily with respect to the motion of the underlying particles, as shown in Figure 2.4. The only constraint is that node movements should not distort the mesh too much

as this leads to computational inaccuracy. Many algorithm exist to implement suitable quality criteria and keep the mesh motion reasonable and to allow the nodes to follow moving particles up to a certain extent [8].

Since the mesh motion and material particle motion are not directly linked, a new unknown is introduced: the relative movement between the ALE mesh and the material domain. This approach is particularly useful in FSI problems: fluid and solid must follow the moving interface between them for physical reasons. Since the solid domain is usually described in a Lagrangian perspective, the solid mesh is kept attached to the FSI interface. However, also the fluid domain must deform to avoid formation of gaps between the meshes. Therefore, in ALE methods the fluid mesh nodes at the interface move with it. Fluid mesh nodes follow the fluid particles sticking to the interface (for viscous flows), while the rest of the fluid mesh is allowed to move in such way that mesh distortions are kept minimal, to preserve computational accuracy [9].



(a) undistorted mesh

(b) distorted mesh

Figure 2.4. ALE mesh

2.2 Domains and interface

Fluid-structure interaction implies that the overall model is determined by models defining the fluid behavior and the solid behavior, briefly described in sections 2.2.2 and 2.2.1. A short overview of beam models is given in section 2.2.3 as it is relevant for the model developed in this work. Finally a formal definition of the interface is given in section 2.2.4, as it is necessary to define suitable coupling conditions at the common boundary of the solid and the fluid.

2.2.1 Fluid domain

An exhaustive description of all possible fluid models is far beyond the scope of this work. A quite general model is the viscous compressible one described by the Navier Stokes Equations (NSE).

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (2.4a)$$

$$\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \otimes \vec{v}) + \nabla p - \nabla \cdot \tau - \rho \vec{g} = 0 \quad (2.4b)$$

$$\frac{\partial}{\partial t} (\rho e_0) + \nabla \cdot (\rho e_0 \vec{v}) + \nabla \cdot (\vec{v} p + \vec{q} - \vec{v} \cdot \tau) - \vec{v} \cdot \rho \vec{g} = 0 \quad (2.4c)$$

where:

- ρ denotes density
- \vec{v} is flow velocity in all dimensions
- p denotes pressure
- $\boldsymbol{\tau}$ is the viscous stress tensor
- \vec{g} represents the sum of all body forces
- e_0 is the total energy per unit mass
- \vec{q} is the heat flux by conduction

They consist in the mass conservation equation (2.4a), the conservation of momentum equation (2.4b) and the energy conservation equation (2.4c). For a Newtonian fluid, the viscous stress tensor is given by:

$$\boldsymbol{\tau} = -\frac{2}{3}\mu (\nabla \cdot \vec{v}) \boldsymbol{I} + 2\mu \boldsymbol{S} \quad (2.5)$$

with μ being the dynamic viscosity and \boldsymbol{S} the rate of deformation tensor (i.e. the symmetric part of the velocity gradient $\nabla \vec{v}$):

$$\boldsymbol{S} = \frac{1}{2} (\nabla \vec{v} + \nabla \vec{v}^T) \quad (2.6)$$

A detailed derivation of such equations and the theory beyond can be found for example in [10] and [11] or in [12].

The set of equations above, even with a Newtonian fluid model, lack some other information in order to form a closed set of Partial Differential Equations (PDE). A conductive heat flux model is needed (e.g. Fourier's Law), the caloric and thermodynamic equations of state have to be chosen, a proper turbulence model if needed (see [12]) and finally, the appropriate initial and boundary conditions for the problem [13] must be defined.

Simplifications can be done to obtain less sophisticated models such as: adiabatic, inviscid, incompressible, and many others. Dimensional Analysis is a powerful tool to determine to what extent some reduced models are meaningful, and it is widely used in fluid dynamics, as described in section 2.3.1. Most CFD software codes allow to set up simulations with the most suitable model which can be coupled with a solid model to build a FSI problem. Some further details are given in section 3.1.

2.2.2 Solid domain

In solid mechanics, particles do not travel as much as they do in fluid dynamic problems, as described in 2.1.2. For this reason, a Lagrangian perspective is generally used.

The de-Saint Venant-Kirchhoff model [14] is very commonly used when describing the movement of a solid: it is also often used in FSI problems as it is capable of handling large deformation. The material is considered:

- *homogeneous*: the material properties do not depend on the position of the particle
- *linear elastic*: the stress-strain relationship is linear
- *isotropic*: the stress-strain relationship is independent from the direction of the load

A general expression of the dynamic equation can be derived from the Virtual Work Principle (VWP) applied to an arbitrary control volume:

$$\frac{\partial^2 \vec{u}}{\partial t^2} = \nabla \cdot \mathbf{T} + \rho \vec{f} \quad (2.7)$$

In equation 2.7:

- ρ : is the material density
- \vec{u} : is the particle displacement
- \mathbf{T} : is the *second Piola-Kirchhoff* stress tensor
- \vec{f} : is the sum of body forces

In order to close the dynamic equation, a constitutive law which must be considered to relate stress and strain:

$$\mathbf{T} = \lambda \mathbf{I} \text{tr} [\boldsymbol{\varepsilon}_G] + 2\mu \boldsymbol{\varepsilon}_G \quad (2.8)$$

where $\boldsymbol{\varepsilon}_G$ is the Green-Lagrange strain tensor:

$$\boldsymbol{\varepsilon}_G = \frac{1}{2} (\mathbf{F}^T \mathbf{F} - \mathbf{I}) \quad (2.9)$$

and \mathbf{F} is the deformation gradient. λ and μ are material properties and are named Lamé constants. These relate to the Young modulus E and the Poisson ratio ν which are more commonly used in practice. The relationship among the various parameters is the following:

$$E = \frac{\mu(3\lambda + 2\mu)}{\lambda + \mu} \quad (2.10)$$

$$\nu = \frac{\lambda}{2(\lambda + \mu)} \quad (2.11)$$

The set of parameters (E, ν) or (λ, μ) , together with the density ρ fully define the material, under the assumptions of linear elasticity, isotropy and homogeneity.

The set of PDEs is completed when suitable initial and boundary conditions are defined.

2.2.3 Models with reduced dimensionality: beams

The equations introduced in section 2.2.2 may be a tough task to solve even of the case of isotropic hyperelasticity, when considering a 3-D domain. Even with today's computers and using finite elements techniques, it is not always feasible or convenient to treat a solid as a three-dimensional continuum. Body with particular geometric features can be seen as lower dimension bodies, with respect to the governing equations [15]. Such bodies are called *beams* (one dimension) , *plates* or *shells* (two dimensions).

The *beam* model splits the description of the geometry into two subproblems:

1. a beam is defined by its *reference line* and the movement (displacement and rotation) of the solid is completely defined by it (see Figure 2.5),

- the beam *cross section* is considered as a whole, its movement depends on the movement of the reference line, stresses are generalized into *resultants* (axial, bending, shear, torsional) which represent the aggregate effect of all of the stresses acting on the cross section. The constitutive properties of the section (axial, shear, torsion and bending stiffness) allow to relate stresses and deformations (by means of VWP) and close the problem.

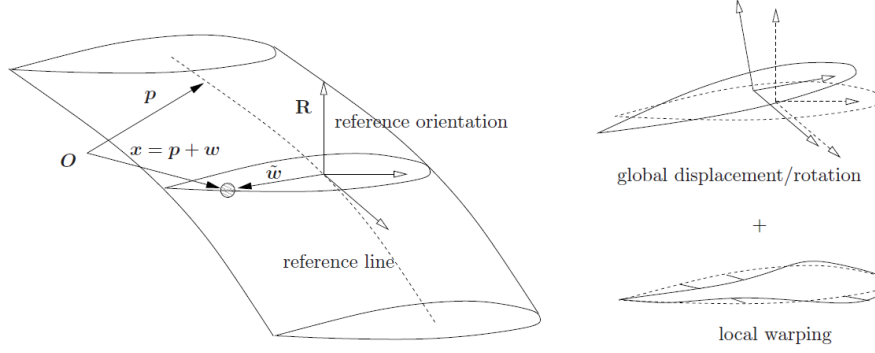


Figure 2.5. beam model, taken from [1]

The beam model can be used to build elements of a Finite Element Method (FEM). For example, the beam element can be modeled by means of a Finite Volume approach, as described in [16], which computes the internal forces as functions of the straining of the reference line and orientation at selected points along the line itself, called evaluation points.

This approach is particularly interesting for FSI problems in which slender structures are involved. A mapping is needed between the fluid-solid interface and the reference line movement, which will be described in.

2.2.4 Interface and interaction

Since FSI problems are centered on the interaction of the fluid and solid domain, their common interface needs to be described properly. A simple representation of the situation at the so called *wet surface* is shown in Figure 2.6. Quantities related to the solid use S subscript, while fluid domain and the interface are labeled with F and FS, respectively.

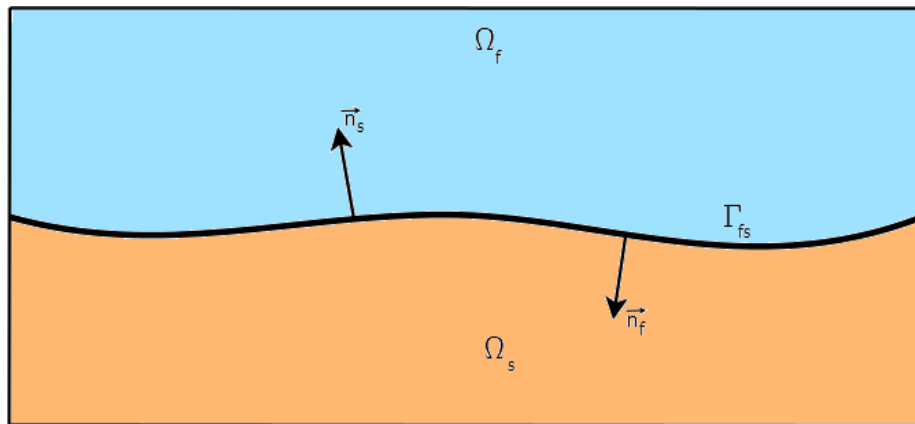


Figure 2.6. fluid solid interface

In order to have a physically correct behavior, some conditions have to be met [?]:

- solid and fluid domains should not overlap nor separate,
- for a viscous fluid model, the flow velocity at the interface must equal the boundary velocity (*no-slip* condition),
- for an inviscid fluid model, only velocity components normal to the wet surface have to be equal to the structural velocity as the fluid may slip freely in tangential direction at any boundary,
- forces exchanged at the interface must at equilibrium.

The first conditions result in the *kinematical requirement* that the displacements of fluid and solid domain, as well as their respective velocities have to be equal at the wet surface (denoted by Γ_{FS}):

$$\Delta \vec{x}_F = \vec{u}_S \quad (2.12)$$

$$\vec{v}_F = \frac{\partial \vec{u}_S}{\partial t} \quad (2.13)$$

The last condition results in the equilibrium requirement. Force vectors are computed from the stresses at the interface and the outward normal vectors of fluid and solid domain, respectively. They have to be equal and opposite leading to the dynamic coupling condition:

$$\sigma_F \cdot \vec{n}_F + \sigma_S \cdot \vec{n}_F = 0 \quad (2.14)$$

$\sigma \in \mathbb{R}^{3 \times 3}$ represents the stress tensor (note that for the fluid it comprises pressure and viscous stresses), while $\vec{n} \in \mathbb{R}$ is the outward normal unit vector.

2.3 Classification of FSI problems

you have been interested in the effect of boundary conditions on the flow. For instance, here is the effect of a cylinder that deviates a uniform flow. In fluid mechanics, we consider solids as boundary conditions only, and not in terms of what they are made of. In solid mechanics, we usually consider fluids just as the cause of a loading at the boundary, a force type boundary condition. These two approaches are very useful, and I extensively used in engineering. For instance, in civil engineering you may find engineers that compute wind loads on a bridge. And then send them to other engineers that will check that this is acceptable in terms of the solid mechanics of the bridge. This is often quite sufficient. We mean situations where you cannot solve these two problems independently. Here schematically, the cylinder is deformed by the flow, which itself is modified by the deformation of the cylinder. This is coupled fluid and solid mechanics. Now, when you think of it, this question of coupling of models is actually quite fundamental.

First, find a way to classify all these couplings. Why? Because the variety seems so large that it does not seem feasible to find a model that is applicable to all of them. The second objective, once we have classified them is to try and build relevant models for these classes.

So when you want to go from considering dimensional quantities to dimensionless ones, what can you do? There is a rather general theorem called the Pi Theorem or the Vaschy-Buckingham Theorem, which tells you how many dimensionless quantities you need to look for. This theorem states that the number of dimensionless quantities, P , is equal to that of the dimensional ones, N , minus R . What is R ? It is the rank of the matrix of dimension exponents. This matrix is formed by the columns of the dimension exponents of all variables,

as you can see here. Remember that the rank of a matrix is a number of independent lines or columns that you can find. Let us give an example

2.3.1 Dimensional analysis

CFR w1-2

we need to classify all these cases of mechanical coupling between fluids and solids. The tool we shall use is called Dimensional Analysis. It is dimensionless in the sense that we do not need any scale of unit to express it. It is just a number. Here, let us just take as a principle that a physical law should only relate dimensionless quantities. There is a rather general theorem called the Pi Theorem or the Vaschy-Buckingham Theorem, which tells you how many dimensionless quantities you need to look for. This theorem states that the number of dimensionless quantities, P , is equal to that of the dimensional ones, N , minus R . What is R ? It is the rank of the matrix of dimension exponents. This matrix is formed by the columns of the dimension exponents of all variables, as you can see here. Let us consider, schematically, the fluid here in blue and a solid in red. To make things simpler, we assume that they stand in separate domain of space and that there is no mass transfer between them. As for the case of drag on a sphere, we now need to specify what quantities we want to use to define our problem and what we are looking for. First, the fluid on the left. Let us say that we're looking for the local velocity U in relation to the coordinate of the point we consider X and the time T . This is also going to depend on the viscosity of the fluid, μ , The density of the fluid ρ and the gravity, G . Also the result is going to be different if I change the size of the domain. So I say the result depends on the size L . Of course, this velocity is also going to depend on some boundary condition. For instance, an upstream flow velocity that I call U_{naught} . This is the list of quantities I'm considering in a given problem in the fluid. Second, the solid on the right. We might want to know the displacement, csi , at a position X , at a time T . It may depend on E , the stiffness of the solid, (I shall come back to this later) on its density, ρ_s and on gravity G . Again, it will also depend on the size. And there is somewhere the magnitude of this displacement that is set, say cdi_{naught} . As you can see here, I'm quite general in stating what the problem is. Still by stating that this is the list of quantities that I want to relate by my physical law, I'm not that general. For instance, I've excluded the temperature. But we have to choose what is the kind of problems that we want to consider. And this is already quite general.

2.3.2 Dimensional analysis in fluid domain

CFR w1-3

We are going to start by something very simple. Doing dimensional analysis separately in the fluid and in the solid. Imagine now that what happens in one domain is totally independent of what happens in the other. For instance, in the fluid. This is what you have done in fluid mechanics when you have ignored all possible influence of what happened inside a solid that bounds the fluid. So, we assume that there exist a physical law that relates the fluid velocity with all the other parameters, namely X , T and so on. This means that the flow is not going to depend on the deformation of the solid, because the stiffness E for instance, is not included in there. This is pure fluid mechanics. Let us do the dimensional analysis of this. Here is a law F between the dimensional variables. There are eight. To use pi theorem, I need to build the matrix of dimension exponents. Here it is. X is the coordinate, so it is a length. T is a time. U is a length per time and so on. As soon as you can put some units on these quantities, you can write the dimension exponents. Now, what is a rank of this matrix? We can find three independent vectors. For instance, here.

And certainly no more than three because the dimension is three. So the rank is all equal to three. I can conclude that we should be looking for 8 minus 3 equals 5 dimensionless parameters. So, let us write the law we are looking for in the form of one depending on only five dimensionless parameters. What are these dimensionless parameters? We know that we should find five independent ones. I can easily start by defining a dimensional velocity by dividing U by U_{naught} . Both are velocities. So the ratio is dimensionless. Second, X divided by L . Third, something I shall explain in a moment. Then, of course, the Reynolds number that combines these four. What else? I haven't used the gravity G so far. So let us use it in a dimensionless number. Here is what is usually called the Froude number combining U_{naught} , G and L . These five members are dimensionless and they are independent. You cannot get one by a combination of the others. Let us go back to the ratio $U_{\text{naught}} T$ over L . As all dimensionless quantity, this one can be understood as the ratio of two dimensional quantities, two lengths, two times. I can write this one as T over T_{fluid} where T_{fluid} equals L over U_{naught} . What is L over U_{naught} ? It is just the time taken by a particle of velocity U_{naught} to travel across the distance L . So T_{fluid} is a time scale associated with convection in the fluid. A very important quantity that we shall use later. At this stage, we have just written down the fact that the dimensionless velocity in the fluid is dependent on a dimensionless coordinate, a dimensionless time, the Reynolds number, the Froude number.

2.3.3 Dimensional analysis in solid domain

Let us now do the same for the solid alone. Now, we look for a relation between all quantities on the solid side. F of $X, T, \text{csi}, E, L, G, \text{tho s chi naught}$, equals zero. I have singled out the displacement, which is unknown. Let us use again the pi theorem. Here is the matrix of the dimension exponents. We have here, too, 8 quantities, a rank of 3, and so 5 dimensionless parameters to find. What are they? Here is a choice. The dimensionless displacement where I've divided csi by the length L . The dimensionless coordinate or dimensionless time, I will discuss just after, and two other dimensionless parameters. The first one is the ratio between the displacement data csi naught and the length scale of the system. We shall call it the displacement number. When large, the displacements are large with regard to the size. This is what we call usually large displacements. The second one combines gravity, density, length and stiffness and I shall call it the elastogravity number. When it is large, it means that the deformations induced by gravity in the solid are large. For instance, in a jelly cake, the shape is really effected by gravity. Let us go back now to the dimensional time that I introduced. I can write this as T over T_{solid} , where T_{solid} is L over a velocity C , and this velocity is square root of E over ρ . What is it? It is actually the scale of elastic wave velocities inside the solid. So T_{solid} is the time that an elastic wave takes to go across the solid.

2.3.4 Dimensional analysis of coupled problems

cfr W1-4

We are now ready to undertake the dimensional analysis of a fully coupled fluid and solid interaction problem. We have done the case of the fluid alone and the case of the solid alone. We're going to use exactly the same method but considering the fluid and the solid, simultaneously. We are back to our full list of parameters that define the problem. Let us discuss a bit what these quantities are. Some of them are only defined on the fluid side, or on the solid side. This is, for instance, the case of the viscosity μ in the fluid or the stiffness E in the solid. Others are common to both domains such as the gravity g or the scale of lengths L . What about our variables of interest, those that we want to relate to the

parameters? I mean, the velocity U or the displacement δ . One of them is defined in the fluid and the other in the solid. But now, we are going to consider that they are related to all the parameters of the problem without separation.

What are these dimensionless quantities in such of problem that mixes fluid and solid? Let us try to give a set of eight independent dimensionless quantities out of the 11 dimensional ones. I'll start with the one I know. U over U_{naught} , x over L , $U_{\text{naught}} t$ over L , the Reynolds number and the Froude number. That makes five. Now, I can also use the ones I know from the solid side, combining the three quantities in a solid, and that gives us the displacement number, δ over L , and the elastogravity number, G . That makes $5 + 2$ equals 7. But from the pi theorem I know I should use eight dimensionless quantities.

It necessarily mixes things from the fluid and the solid side otherwise I would have found it before when doing the uncoupled case. So what is it? What can we imagine as the dimensionless quantity combining fluid and solid dimensional once.

Mass number

The simplest one is the ratio of the two densities. Let us call it the Mass Number, M . This seems a very good choice because it simply tells you that it is different for a solid to interact with air or with water. In the hard-disk drive example, M is the order of 1, air, over 10 to the 4, metal, and so M is of the order of 10 to the minus 4. Conversely, for the dolphin skin, both media have about the same density, and M is the order of 1.

Reduced velocity

Here is another possible choice, the reduced velocity. It is the ratio between our free velocity, U_{naught} , and the velocity of elastic waves in a solid, c . This also seems a good idea because it contains information on the way the two dynamics are related. It would be quite different between two examples I considered before. The inflatable dam and the dolphin's skin. [MUSIC] As possible new dimensionless parameters I have proposed the ratio of two densities, that was the mass number and the ratio of two velocities, that was the reduced velocity.

Cauchy number

I can also imagine something combining stresses or stiffness. This here is the Cauchy number. What does it mean? It is the ratio between the fluid loading quantified by the dynamic pressure over a unit square and the stiffness of the solid E . The higher it is, the more the solid is elastically deformed by the flow.

These three are actually the most important ones, and are used a lot. Which one should you choose for your problem? Well as I said before, there is no good choice of dimensionless numbers. But there are efficient choices, that would be more helpful in solving a given problem.

Figure 2.7.



Figure 2.7. First Figure

Chapter 3

Computational aspects of Fluid-Structure Interaction problems

Subsequent to the basic physics of FSI problems drawn up in the last chapter, this section goes into detail on computational treatment, which also allows for FSI techniques to be categorized. Two fundamentally different classes of procedures for solving FSI simulations have been established, namely monolithic and partitioned approaches. They are discussed in Section 3.1. Since this thesis focuses on the latter kind, a characterization of weak and strong coupling in partitioned approaches is described in Section 3.2. Subsequently, in Section 3.3, a different way of categorizing FSI techniques is presented, which is based on conforming or non-conforming mesh treatment of respective solver strategies. The chapter is closed by Section 3.4, having a look at the added mass effect (AME) - a stability issue arising from problems with strong interaction in partitioned FSI simulations.

3.1 Monolithic and Partitioned Approach

First of all, I want to point out that the terms monolithic and partitioned are interpreted differently throughout FSI literature. However, in this thesis, they are only used in the hereafter explained sense. Again, let the name fluid-structure interaction serve as a motivation for this section: On the one hand, it implies a single (but coupled) physical problem while on the other hand, its clear multiphysical characteristic is emphasized. This is also reflected in the monolithic and partitioned approaches, respectively. However, monolithic and partitioned approaches should not be interpreted as solely oppositional. There exist solver strategies for which the boundaries between the two approaches blur. Monolithically treating both fluid and solid domain implies that they are solved simultaneously. This means that one multiphysics solver deals with a single system of equations describing fluid, solid and their coupling. Figure 3.1 depicts this strategy schematically. Such monolithic solvers are designed specifically for the sole purpose of solving FSI problems. Therefore, a high level of specialization can be realized. Simultaneously treating both flow and structural equations often results in good numerical stability of the calculations. Furthermore, monolithic approaches solve the system of equations exactly, meaning there are no errors (other than those which are inherent to numerical techniques) introduced by this form of numerical treatment. However, development of such solvers from scratch requires a lot of coding work and is often cumbersome ([8], [38], [20], [18]). In contrast, partitioned approaches make use of existing single-physics solvers. The FSI problem is split into a fluid and solid problem, which are both treated by their respective solvers separately, while a third software module, the coupling component,

incorporates the interaction aspects. It communicates forces or stresses (dynamic data) calculated by the fluid solver at the wet surface to the solid component and exchanges displacements or velocities (kinematic data) computed by the solid solver at the interface to the fluid component in return ([18], [38]). A schematic sketch of this situation is shown in Figure 3.2. More detailed and practical explanations about the coupling component are given in Section 4.1. For now it is sufficient to know that the coupling component exchanges kinematic and dynamic data between the single-physics solvers in order to preserve the coupled nature of the overall problem. In the end, fluid and structural solutions together yield the FSI solution. By analogy with the monolithic approach, it is graphically depicted in Figure 3.3. A big advantage of this approach is that existing solvers for the fluid and solid problem can be reused, ranging from commercial to academic and open-source codes. Especially in the commercial sector, these are often highly elaborate solvers with decades of experience in their particular single-physics fields. They typically support very sophisticated solution techniques. Also, those solvers are usually well-validated and compared to monolithic procedures the programming efforts are lower for partitioned approaches, as only the coupling of the existing solvers has to be implemented rather than the solvers themselves. Nevertheless, these advanced solvers can only be of good use for FSI simulations if the coupling component is sufficiently precise ([8], [38], [20]).

3.2 Coupling Strategies

Partitioned strategies for solving FSI problems can be subclassed into weakly and strongly coupled approaches. They are also referred to as explicit and implicit methods in this thesis. Note that this nomenclature is in no way fully consistent throughout FSI literature. However, all usage of these terms in this thesis is limited to the sense of the explanations given in this section. The distinction between explicit and implicit techniques is based upon the question, how often solutions for the fluid and solid subproblem are computed within one time step and also, how frequently the relevant kinematic and dynamic quantities are exchanged. For weakly coupled strategies solving is done a certain, fixed number of times (often only once) per time step and data may not even be communicated after each discrete time instance. In general, this is not sufficient to regain the monolithic ("exact") solution of the FSI problem as the coupling conditions are not enforced within each time step. Thus, no balance between fluid and structural domain with respect to energy, forces and displacements at the interface can be guaranteed ([8], [18], [38], [2]). However, this coupling strategy can still yield good results if the interaction between fluid and solid is rather weak (further explanations about the strength of the interaction follow in Section 3.4). E.g. in aeroelastic simulations, where small displacements of the structure appear within single time steps, the flow field is influenced by the structural displacements only to a little extent ([13], [8], [2]). In contrast, strongly coupled strategies make use of subiterations possibly resulting in multiple computations of the separate solvers and exchanges of the interface coupling quantities (as a reminder, see Figure 3.2) per physical time step. However, acceleration techniques are necessary to converge the underlying coupling equation system. The coupling conditions at the wet surface are enforced in each time step up to a convergence criterion. If the criterion is not met sufficiently, another subiteration within the same time instance is calculated. Therefore, the solution can approximate the monolithic solution to an arbitrary accuracy as the convergence criterion can be chosen as strict as needed. Such a method is in general applicable to both FSI problems, which can be solved by weakly coupled approaches and those, for which explicit procedures fail due to dominant interaction. However, strongly coupled algorithms are usually used in the latter case - when weak coupling reaches its

limits - since the implicit approach requires more computational effort ([18], [38], [2]). Figure 3.4 sums up the categorization of different coupling techniques mentioned in this and the previous section. They are ordered with respect to stability and programming effort.

3.3 Interface Mesh

In this section, FSI methods are classified by means of two different mesh treatment procedures: conforming or non-conforming techniques. The basic question is, whether fluid and solid mesh need to align with each other at the FSI interface or not. Unless stated otherwise, the explanations of this section are taken from [20]. Note that some aspects of conforming mesh methods are already included in the previous sections without explicitly mentioning so, in order to develop a better understanding of partitioned FSI simulations. Conforming mesh methods usually consist of three major subtasks, namely computation in the fluid and solid domain, as well as interface and mesh movement. They require both fluid and structural meshes to conform to the wet surface, because the coupling conditions are applied via the interface as physical boundary conditions for the respective domains. This does not necessarily imply node-to-node matching of fluid and structure meshes at the interface. This must hold for all time instances, which means that both fluid and structural grids need to be moved in case deformations of the solid appear. This is a simple task for the solid mesh, since it is usually expressed in a Lagrangian fashion anyway. However, as a typical Eulerian fluid mesh would not follow the interface motion, the necessity of the ALE method as discussed in Section 2.2.3 becomes apparent. Also, mesh smoothing techniques need to be introduced in order to prevent quality losses of the fluid mesh in terms of distorted elements. These irregularities lead to accuracy loss in simulations. In Figure 3.5, a conforming mesh is shown at two different points in time. At the first instance (Figure 3.5a) the solid is undeformed and therefore, also the fluid mesh remains in its initial configuration. In contrast, at the second point in time (Figure 3.5b) the solid is deformed and the fluid mesh conforms to the displaced wet surface. Consequently, also mesh smoothing is applied. There is a wide variety of such mesh updating procedures. Some common techniques compute the mesh movement by considering mesh edges as springs ((torsional) spring analogy), solving the Laplace equation or solving a pseudo-structural system of equations (see e.g. [18], [43], [20] and their respective references for further explanations of these techniques). Conforming mesh strategies are widely, but not exclusively used in partitioned FSI approaches. Furthermore, they typically also utilize the ALE method ([20]). In contrast, in non-conforming mesh strategies all interface conditions are directly imposed as constraints on the flow and structural governing equations. Therefore, it is possible to use non-conforming meshes for fluid and solid domains as they remain geometrically independent from each other. Thus, also mesh smoothing techniques are obsolete [43]. Figure 3.6 depicts such a situation. By analogy with Figure 3.5, again the initial configuration (Figure 3.6a) and an instance when the solid is deformed (Figure 3.6b) are shown. It is clearly visible that the fluid mesh does not conform to the wet surface as all nodes stay at the same position regardless of the solid deformation. This approach is mostly used in immersed methods. The considerations in this section are limited to them, as they are also very common for FSI simulations. Coupling is imposed via additional force-equivalent terms appearing in the model equations of the fluid, enforcing the kinematic and dynamic conditions. These FSI forces are computed from the structural model, which is dealt with separately together with tracking the position of the interface. The forces represent the effects of a boundary or body being immersed in the fluid domain (leading to immersed boundary and immersed domain methods). A purely Eulerian mesh can be applied to the whole computational

domain for solving the fluid equations, since the force-equivalent terms are dynamically added in a spatially specific manner to those locations, which are currently occupied by the structure. After solving the fluid equations, forces exerted on the solid at the wet surface are computed and used as input for the structural solver, which still employs a Lagrangian mesh. Subsequently, the deformation of the solid material is calculated and the displacement of the FSI interface is fed back to the fluid model in form of updated force-equivalent terms ([31], [20], [43]).

3.4 Stability: Added Mass Effect

To conclude this chapter about computational aspects of FSI simulations, the AME is briefly described. Explanations of this effect can be found in a great variety throughout FSI literature, typically explicated for specific solver strategies or flow regimes (see e.g. [5], [42], [15], [2]). Therefore, in the scope of this thesis only a short phenomenological introduction to the concept of added mass and numerical problems arising from it is given. However, this suffices to focus on both weakly and strongly coupled partitioned approaches, which are practically relevant in this thesis. The AME is inherent to partitioned FSI approaches as the single-physics fields are not continuously coupled but interaction only occurs at a finite number of discrete time instances, when coupling quantities are exchanged. As already mentioned in Section 2.3.3, there can be no gaps between structure and fluid. Also their respective particles cannot occupy the same spatial locations simultaneously. Thus, if the solid is moved, also fluid particles move. Changing the state of motion of the structural component consequently requires taking into account inertial effects not only of the solid itself, but also of the surrounding fluid, which artificially rests for the span of a single structure solver time step. In more descriptive words: Moving the solid also implies moving fluid particles close to the solid. Therefore, the structure behaves more inert due to artificially added mass ([42], [2]). Since inertia is dependent on mass and therefore density, the ([42], [5]): This ratio is often used to describe how strong the interaction between solid and fluid is. For cases, in influence the FSI problem (weak interaction). But as fluid and structure densities approach each other interaction) and imposes stability limits on partitioned solution techniques ([5], [42], [2]). Note that the AME is not only governed by the density ratio of Equation 3.1 but also by geometric properties of the problem, the stiffness of the solid ([5]) and the speed of sound in the flow domain ([42]). Nonetheless, for the sake of simplicity and intuitiveness, explanations in this thesis are mostly limited to the density ratio. In general, the AME is of bigger concern for incompressible flows than for compressible regimes. From a physical point of view, deformations of the structural domain can be interpreted as perturbations for the flow field. In compressible flows the speed of sound (speed at which perturbations propagate through the flow) is finitely large. Thus, the influence of a geometrical change of the fluid domain caused by deformations of the solid is locally limited during a certain period of time. In contrast, in incompressible flows the speed of sound is infinitely large, hence all perturbations propagate through the flow without time delay. Therefore, regardless of how much time has passed since a perturbation, the whole flow field is directly affected ([42], [5]). In the following it is assumed that a weakly coupled algorithm allows computation of the fluid and solid solution only once per time step. In addition, coupling data is also exchanged once per time instance. In contrast, a strongly coupled solver does the same at least twice per time increment (for a reminder see Section 3.2 and compare Figure 3.4). As can be shown, in the compressible case a more dominant AME can be compensated for by reducing the time step size of strongly coupled, partitioned solution algorithms. This however, does not hold for the incompressible regime, where even in the

limit of vanishing time step size strongly coupled, partitioned algorithms might fail ([42]). These observations are consistent with the above mentioned physical explanation. First of all, considering compressible flows, indeed, the lack of repeated subiterations in weakly coupled partitioned techniques leads to a strict limit for the density ratio (of Equation 3.1) due to the fact that the interface conditions are not enforced and energy balance at the wet surface is generally not given. If that limit is exceeded the algorithm fails due to instability ([2]). Likewise, in such a case a strongly coupled partitioned algorithm converges slowly, resulting in possibly many necessary subiterations, which is computationally costly. Yet it does not become unstable, given that the time step size is chosen sufficiently small. Reducing the time step size to an arbitrarily small extent cannot stabilize a weakly coupled approach if the stability criterion on the density ratio is not met ([15]). Conversely, the convergence rate of strongly coupled algorithms increases by the same factor, by which the time step size decreases, meaning that in the limit of vanishing time step size the monolithic solution is obtained ([5], [42]). In the incompressible case however, a strict stability limit exists for both weakly and strongly coupled algorithms. It is independent of the size of time increments¹. Furthermore, in order for an implicit method to achieve the monolithic solution (assuming its convergence is given, i.e. the before mentioned stability limit is not exceeded) the number of subiterations must be increased as time step size decreases ([15], [42]).

Chapter 4

Software Packages used in this work

Lorem ipsum dolor sit amet, consectetur adipiscing elit, sed eiusmod tempor incididunt ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur. Quis aute iure reprehenderit in voluptate velit esse cillum dolore eu fugiat nulla pariatur.

Chapter 5

MBDyn Adapter and its integration

Lorem ipsum dolor sit amet, consectetur adipiscing elit, sed eiusmod tempor incididunt ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur. Quis aute iure reprehenderit in voluptate velit esse cillum dolore eu fugiat nulla pariatur.

Chapter 6

Validation Test-cases

Lorem ipsum dolor sit amet, consectetur adipiscing elit, sed eiusmod tempor incididunt ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur. Quis aute iure reprehenderit in voluptate velit esse cillum dolore eu fugiat nulla pariatur.

Conclusions

27

[illegible]

ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur. Quis aute iure reprehenderit in voluptate velit esse cillum dolore eu fugiat nulla pariatur. Excepteur sint obcaecat cupiditat non proident, sunt in culpa qui officia deserunt mollit anim id est laborum. Lorem ipsum dolor sit amet, consectetur adipisci elit, sed eiusmod tempor incidunt ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur. Quis aute iure reprehenderit in voluptate velit esse cillum dolore eu fugiat nulla pariatur. Excepteur sint obcaecat cupiditat non proident, sunt in culpa qui officia deserunt mollit anim id est laborum. Lorem ipsum dolor sit amet, consectetur adipisci elit, sed eiusmod tempor incidunt ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur. Quis aute iure reprehenderit in voluptate velit esse cillum dolore eu fugiat nulla pariatur. Excepteur sint obcaecat cupiditat non proident, sunt in culpa qui officia deserunt mollit anim id est laborum. Lorem ipsum dolor sit amet, consectetur adipisci elit, sed eiusmod tempor incidunt ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur. Quis aute iure reprehenderit in voluptate velit esse cillum dolore eu fugiat nulla pariatur. Excepteur sint obcaecat cupiditat non proident, sunt in culpa qui officia deserunt mollit anim id est laborum. Lorem ipsum dolor sit amet, consectetur adipisci elit, sed eiusmod tempor incidunt ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur.

First Appendix

31

[illegible]

ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur. Quis aute iure reprehenderit in voluptate velit esse cillum dolore eu fugiat nulla pariatur. Excepteur sint obcaecat cupiditat non proident, sunt in culpa qui officia deserunt mollit anim id est laborum. Lorem ipsum dolor sit amet, consectetur adipisci elit, sed eiusmod tempor incidunt ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur. Quis aute iure reprehenderit in voluptate velit esse cillum dolore eu fugiat nulla pariatur. Excepteur sint obcaecat cupiditat non proident, sunt in culpa qui officia deserunt mollit anim id est laborum. Lorem ipsum dolor sit amet, consectetur adipisci elit, sed eiusmod tempor incidunt ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur. Quis aute iure reprehenderit in voluptate velit esse cillum dolore eu fugiat nulla pariatur. Excepteur sint obcaecat cupiditat non proident, sunt in culpa qui officia deserunt mollit anim id est laborum.obcaecat cupiditat non proident, sunt in culpa qui officia deserunt mollit anim id est laborum. Lorem ipsum dolor sit amet, consectetur adipisci elit, sed eiusmod tempor incidunt ut labore et dolore magna aliqua. Ut enim ad minim veniam, quis nostrum exercitationem ullam corporis suscipit laboriosam, nisi ut aliquid ex ea commodi consequatur.

Acronyms

FSI	Fluid-Structure Interaction
MBDyn	MultiBody Dynamics analysis software
preCICE	precise Code Interaction Coupling Environment
ALE	arbitrary Lagrangian-Eulerian
CFD	Computational Fluid Dynamics
CSM	Computational Solid Mechanics
NSE	Navier Stokes Equations
PDE	Partial Differential Equations
VWP	Virtual Work Principle
FEM	Finite Element Method

Bibliography

- [1] G. L. Ghiringhelli, P. Masarati, M. Morandini, and D. Muffo, “Integrated aeroservoelastic analysis of induced strain rotor blades,” *Mechanics of Advanced Materials and Structures*, vol. 15, no. 3-4, pp. 291–306, 2008.
- [2] R. C. Batra, *Elements of continuum mechanics*. Aiaa, 2006.
- [3] J. Cheng, G. Zhao, Z. Jia, Y. Chen, S. Wang, and W. Wen, “Sliding free lagrangian-eulerian finite element method,” in *International Conference on Computational Science*, 2006.
- [4] J. Donea, A. Huerta, J.-P. Ponthot, and A. Rodríguez-Ferran, “Arbitrary lagrangian-eulerian methods,” *Encyclopedia of Computational Mechanics Second Edition*, pp. 1–23, 2017.
- [5] J. T. Xing, “Chapter 3 - fundamentals of continuum mechanics,” in *Fluid-Solid Interaction Dynamics*, J. T. Xing, Ed. Academic Press, 2019, pp. 57 – 101. [Online]. Available: <http://www.sciencedirect.com/science/article/pii/B9780128193525000033>
- [6] S. Lipton, J. A. Evans, Y. Bazilevs, T. Elguedj, and T. J. Hughes, “Robustness of isogeometric structural discretizations under severe mesh distortion,” *Computer Methods in Applied Mechanics and Engineering*, vol. 199, no. 5-8, pp. 357–373, 2010.
- [7] A. Bertram, *Elasticity and plasticity of large deformations*. Springer, 2012.
- [8] A. De Boer, M. Van der Schoot, and H. Bijl, “Mesh deformation based on radial basis function interpolation,” *Computers & structures*, vol. 85, no. 11-14, pp. 784–795, 2007.
- [9] E. Ramm and W. Wall, “Fluid-structure interaction based upon a stabilized (ale) finite element method,” in *4th World Congress on Computational Mechanics: New Trends and Applications*, CIMNE, Barcelona, 1998, pp. 1–20.
- [10] L. Quartapelle and F. Auteri, *Fluidodinamica incompressibile*. Casa editrice ambrosiana, 2013.
- [11] —, *Fluidodinamica comprimibile*. Casa editrice ambrosiana, 2013.
- [12] S. B. Pope, “Turbulent flows,” 2001.
- [13] G. Galdi, *An introduction to the mathematical theory of the Navier-Stokes equations: Steady-state problems*. Springer Science & Business Media, 2011.
- [14] R. W. Ogden, *Non-linear elastic deformations*. Courier Corporation, 1997.
- [15] K. D. Hjelmstad, *Fundamentals of structural mechanics*. Springer Science & Business Media, 2007.

- [16] G. L. Ghiringhelli, P. Masarati, and P. Mantegazza, “Multibody implementation of finite volume c beams,” *AIAA journal*, vol. 38, no. 1, pp. 131–138, 2000.