Lehrstuhl für
Strömungsmechanik
Friedrich-Alexander-
Universität
Erlangen-Nürnberg

MASTER THESIS

Dynamical calculation of under-relaxation factor in a partitioned,implicitly coupled fluid-structure interaction solver

NARAYANAN ACHUTHAN

Erlangen, May 24, 2018

Examiner:	[YOUR EXAMINER (probably Prof. Dr. Felix Freiling)]
Advisor:	[YOUR ADVISOR]

Eidesstattliche Erklärung / Statutory Declaration

Hiermit versichere ich eidesstattlich, dass die vorliegende Arbeit von mir selbständig, ohne Hilfe Dritter und ausschließlich unter Verwendung der angegebenen Quellen angefertigt wurde. Alle Stellen, die wörtlich oder sinngemäß aus den Quellen entnommen sind, habe ich als solche kenntlich gemacht. Die Arbeit wurde bisher in gleicher oder ähnlicher Form keiner anderen Prüfungsbehörde vorgelegt.

I hereby declare formally that I have developed and written the enclosed thesis entirely by myself and have not used sources or means without declaration in the text. Any thoughts or quotations which were inferred from the sources are marked as such. This thesis was not submitted in the same or a substantially similar version to any other authority to achieve an academic grading.

Der Friedrich-Alexander-Universität, vertreten durch den Lehrstuhl für Informatik 1, wird für Zwecke der Forschung und Lehre ein einfaches, kostenloses, zeitlich und örtlich unbeschränktes Nutzungsrecht an den Arbeitsergebnissen der Arbeit einschließlich etwaiger Schutz- und Urheberrechte eingeräumt.

Erlangen, May 24, 2018

[YOUR NAME]

Abstract

The presented numerical study focuses on dynamical calculation of the under-relaxation factor during each sub-iteration step of the *Fluid-Structure Interaction* solver using adaptive scehemes. The adaptive schemes presented in the study are *Aitken's* $\triangle^2$ *method* and *steepest descent method*. The mentioned schemes have been found to be efficient, yet easy to implement. The implemented schemes have been validated by a numerical simulation of flow around an elastically mounted circular cylinder at a Reynolds number of 200. (Cite Zhou)

The calculations were performed on a 2-D 0-type curvilinear orthogonal grids containing a total of 120x100 control volumes.The FSI simulations were performed using a *semi implicit predictor-corrector scheme* for fluid-structure coupling.The *semi implicit predictor-corrector scheme* is a strong coupling scheme between flow and structural solver, while also maintaining the explicit time marching schemes. The simulations were carried out for different reduced damping coefficients (Sg) and for a mass ratio (M^*) of 1. These cases were simulated with constant under-relaxation factor, and with dynamic under-relaxation factor using aitken's $\triangle^2$ method and steepest descent methods. The results were compared and validated with (Breuer and Muensch) and (zhou),the results were in good agreement with these established numerical data. Average time taken for sub-iterations within the time steps were calculated, *Aitken's* $\triangle^2$ *method* was observed to be more efficient in accelerating the convergence.

CONTENTS

1 INTRODUCTION

Some general information on the context and setting.

1.1 Motivation

Specific motivation for the problem at hand.

1.2 Task

Concrete task to be solved.

1.3 Related Work

Other relevant academic work and how it differs from this work, for example **?**] and **?**]. Distinguish between “textual” citation, as shown in **?**], and “parenthesis” citation [**?**].

1.4 Results

What has been achieved in this work?

1.5 Outline

How is the thesis structured and why?

1.6 Acknowledgments

A big thank you for the support to . . .

2
THEORETICAL BACKGROUND

2.1 Fluid-Structure Interaction

Fluid-Structure Interaction (FSI) studies the interaction between a structure (solid) and a fluid flow (liquid or gas) around it. It is a multi-physics problem which has large interest in diversified fields such as mechanical engineering (e.g. airfoils), civil engineering (e.g. towers) or medicine technique (e.g. artificial heart valves). Based on the response of structure and fluid fields, it is classified as one-way or two-way fluid-structure interaction problem. If the structural displacement/deformation does not influence the flow fluid or vice versa, then the FSI system is termed to be one-way fluid-structure interaction system. Conversely if the fluid flow and the displacement or deformation of the structure have significant influence on each other then the FSI system is termed as two-way fluid-structure interaction problem. Figure 2.1 represents a typical domain of a fluid-structure interaction problem, Ω refers to the common domain, Ω_f the fluid domain and Ω_s the structure domain. Figure 2.2a and 2.2b represents one-way and two-way FSI fluid-structure interaction problems respectively.

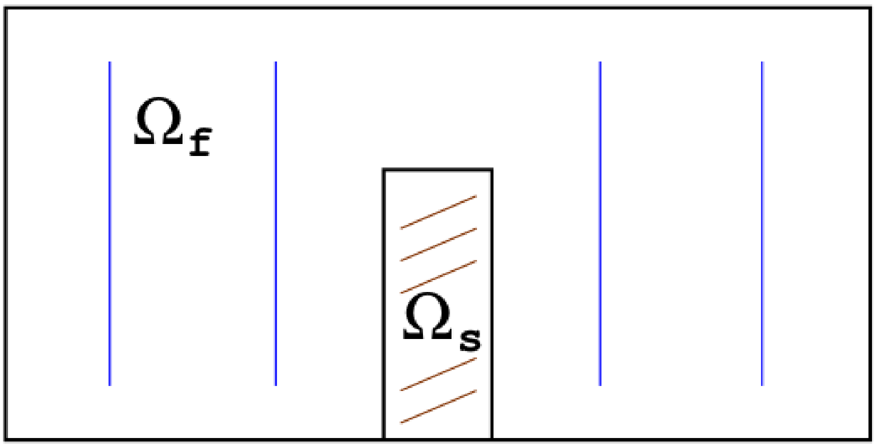
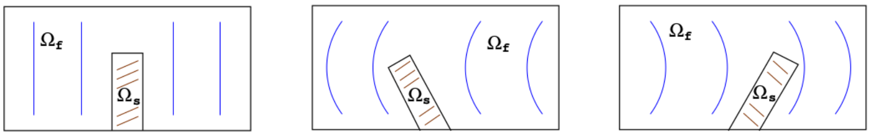
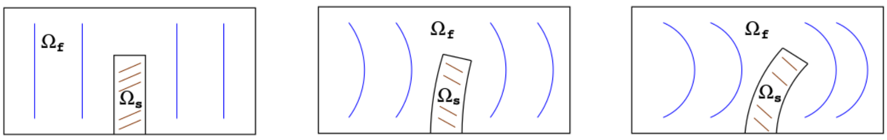


Figure 2.1: Representational figure of Fluid-Structure Interaction domain,taken from Richter [9]



(a) Structure domain influencing the fluid domain



(b) Both structure and fluid domains influencing each other

Figure 2.2: Representational figure of one-way 2.2a, and two-way 2.2b fluid-structure interaction domains,taken from Richter [9]

2.1.1 Numerical simulation strategies of FSI problems

Numerical simulation of FSI involves solving set of differential equations and corresponding boundary conditions for fluid and structural fields respectively. Suitable interface conditions needs to be defined so as the structural and fluid domains are well distinguished. There are different methodologies implemented to solve the FSI problem and are well documented in the literature. An overview of the numerical solution procedure for FSI problems are presented below.

The FSI problem is classified based on solution approaches and on the treatment of mesh handling techniques as represented below. A brief overview of these methods are presented subsequently.

- Classification based on numerical solution approaches
  - Monolithic solver
  - Partitioned solver
    - Explicit coupling
    - Implicit coupling
- Classification based on meshing strategies
  - Conforming mesh
  - Non-conforming mesh

Classification based on numerical solution approaches

This classification is based on how the fluid and structural fields are getting solved. In *monolithic solvers* both the fluid and structural domains are expressed by single set of equations, whereas in *partitioned solver approach* structural fields and fluid fields are solved separately and additionally requires coupling of the distinct solvers. A brief introduction to these methods are represented below.

Monolithic solver approach

The equations governing the fluid flow and the displacement/deformation of the structure are represented by single set of equations which are then solved simultaneously by an unified algorithm, within a single solver framework Becker et al. [1], Hübner et al. [5], Michler et al. [6], Richter [9]. The central idea of monolithic solvers is to represent the interface by an homogeneous discretization, thus maintaining the conservation properties at the interface Michler et al. [6] Van Brummelen et al. [10]. Although monolithic approach gives strong coupling between the fields, it is commonly considered to be impractical for real world applications. It also demands enormous computational power for solving such a large system of equations since it has to incorporate the behavior of both fluid flow and solid structure. Figure 2.3 represents a schematic of the monolithic and partitioned solver approaches in solving FSI problems.

Partitioned solver approach

Another approach to fluid-structure interaction is to use two distinct solvers to model both fluid and solid domains. This technique allows the coupling of the fluid and solid solution

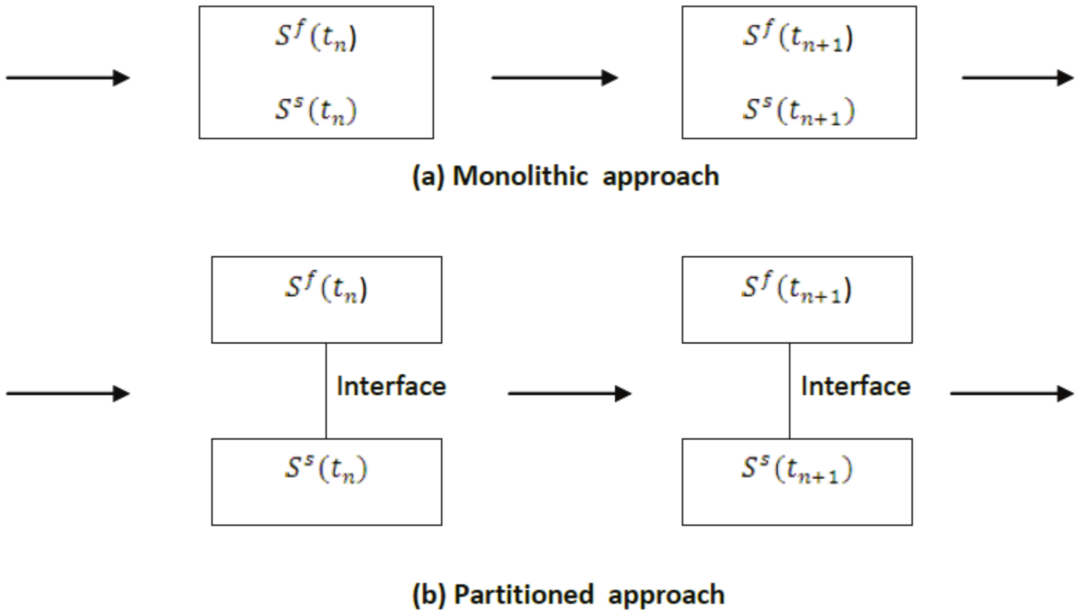


Figure 2.3: Schematic representation of Monolithic and Partitioned solution approaches. S^f represents the fluid solver and S^s represents the structural solver for two successive time steps current (t_n) and next (t_{n+1}). Figure taken from Hou et al. [4]

by maintaining suitable coupling conditions. The interfacial conditions are used explicitly to communicate information between the fluid and structure solutions.

Some coupling algorithms were suggested by various studies (Farhat and Lesoinne [2], Felippa et al. [3], Piperno et al. [8]) which allows for reuse of existing codes that have been developed for each field. This approach is very robust and can be used for wide variety of applications. The major disadvantage of this approach is that, the interface location that divides the fluid and the structure domains is not known a priori and usually changes in time. Thus, the partitioned approach requires tracking of the new interface location and its related quantities, which can be cumbersome and error-prone. The interface coupling conditions that are as stated below, have to be satisfied in order to have a stable solution.

- Kinematic coupling condition: The displacements, velocities and accelerations of the sub-zones have to be equal at the interface at any point in time.

  $\psi_\Gamma^{CFD}(t) = \psi_\Gamma^{CSD}(t)$, $\dot{\psi}_\Gamma^{CFD}(t) = \dot{\psi}_\Gamma^{CSD}(t)$, $\ddot{\psi}_\Gamma^{CFD}(t) = \ddot{\psi}_\Gamma^{CSD}(t)$

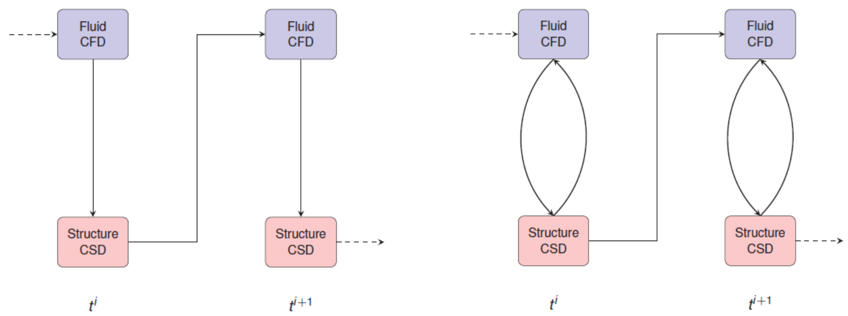
- Dynamic coupling condition: Conservation of the dynamic equilibrium of all forces at the interface needs to be satisfied. (Action and reaction forces must cancel out each other)

  $f_\Gamma^{CFD}(t) = -f_\Gamma^{CSD}(t)$

Depending on the influence of the structure movement on the fluid field the FSI problems are further subdivided into two following categories:

- Explicit/Weakly coupled: In an explicitly coupled algorithm, the equations of fluid mechanics, structural mechanics and the relative mesh movement are solved sequentially. Initially the governing equation of fluid is solved with the velocity boundary condition derived from the structural displacement. The structural mechanics equations are solved next with the forces obtained from the fluid solver. Finally the grid is adapted based on the structural displacement. This coupling scheme is termed as weak coupling, as there is no sub iteration between the two sub zones.
- Implicit / Strongly coupled: In a strongly/implicitly coupled system, sub iterations between both the fluid and structural solvers are carried out at every time step till convergence is achieved. The convergence is determined by maintaining a small enough structural residuum. Detailed explanation on this, is presented in the subsequent section. This approach is implicit in nature and more robust, however requires additional computation time.

Figure 2.4 represents the schematic representation of explicit and implicitly coupled, partitioned approach to solving fluid structure interaction. In the presented study *semi-implicit predictor-corrector* coupling scheme is used.



(a) Explicit coupling **(b)** Implicit coupling

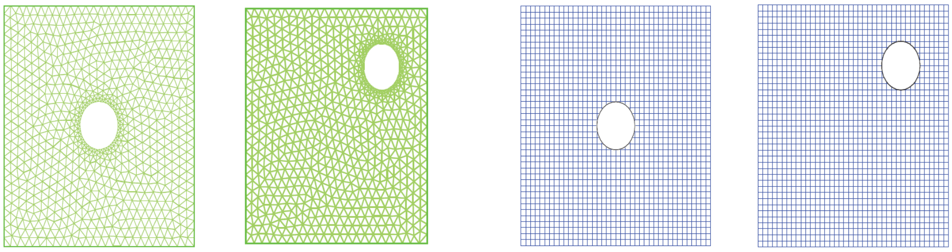
Figure 2.4: Representational sketch of explicit coupling 2.4a, and implicit coupling 2.4b approaches to solving partitioned, fluid-structure interaction problems,taken from Münsch [7]

Classification based on meshing strategies

The FSI problem is further classified based on the meshing strategies implemented to couple the fluid and structure domains. They are classified into two methods:*conforming mesh* and *non-conforming mesh*

- Conforming mesh method: In conforming mesh methods, the interface governing the fluid and structural domains is resolved such that the node connectivity is maintained between the domains. In this method, the interface conditions is considered as physical boundary conditions, which treat the interface location as part of the solution. Owing to the movement and/or deformation of the solid structure,re-meshing (or mesh-update) is required.

- Non-conforming mesh method: The non-conforming mesh methods treat the interface location as constraints imposed on the model equations so that non-conforming meshes (node-connectivity need not be maintained between the domains) can be employed. As a result, the fluid and solid equations can be conveniently solved independently from each other with their respective grids, and re-meshing is not necessary. The distinction between these two types of meshes can be observed in figure 2.5, where a solid body (a sphere) is moving in a fluid domain.



(a) Conforming mesh method representation at time t_n and t_{n+1} **(b)** Non-conforming mesh method representation at time t_n and t_{n+1}

Figure 2.5: Examples of Conforming mesh 2.5a and Non-conforming mesh 2.5b methods, taken from Hou et al. [4].

2.2 Governing equations of fluid flow and structural deformations

In Fluid-Structure Interaction problems, the solid moves through the fluid domain due to different forces or excitation. The computation domain changes with time and must be considered during the simulation. An approach which aims to solve this particular problem is Arbitrary Eulerian Lagrangian formulation, popularly referred to ALE formulation. The governing equation for conservation of mass and momentum are reformulated for a moving grid which are time dependent control volume and surface integrals. The governing equations for a Newtonian incompressible fluid in ALE formulation is given as follows.

$$\frac{d}{dt}(\int_{V(t)} \rho) + \int_{S(t)} \rho(U - U_g).n = 0 F_D = 6\pi\mu a u \tag{2.1}$$

$$\frac{d}{dt}\int_{V(t)} \rho + \int_{S(t)} \rho(U - U_g).n = 0 \tag{2.2}$$

3 Implementation

Some complex code is shown in Figure 3.1.

```
static int __init serpent_init(void)
{
    u64 xcr0;
    if (!cpu_has_avx || !cpu_has_osxsave) {
        printk(KERN_INFO "AVX instructions are not detected.\n");
        return -ENODEV;
    }
    xcr0 = xgetbv(XCR_XFEATURE_ENABLED_MASK);
    if ((xcr0 & (XSTATE_SSE | XSTATE_YMM)) != (XSTATE_SSE | XSTATE_YMM)) {
        printk(KERN_INFO "AVX detected but unusable.\n");
        return -ENODEV;
    }
    return crypto_register_algs(serpent_algs, ARRAY_SIZE(serpent_algs));
}

static void __exit serpent_exit(void)
{
    crypto_unregister_algs(serpent_algs, ARRAY_SIZE(serpent_algs));
}

module_init(serpent_init);
module_exit(serpent_exit);

MODULE_DESCRIPTION("Serpent Cipher Algorithm, AVX optimized");
MODULE_LICENSE("GPL");
MODULE_ALIAS("serpent");
```

Figure 3.1: Serpent AVX module initialization

4 EVALUATION

5 CONCLUSION AND FUTURE WORK

In this chapter we want to draw conclusions about the work, which has been done during this thesis.

5 BIBLIOGRAPHY

[1] P. Becker, S. R. Idelsohn, and E. Oñate. A unified monolithic approach for multi-fluid flows and fluid–structure interaction using the particle finite element method with fixed mesh. *Computational Mechanics*, 55(6):1091–1104, dec 2014. doi: 10.1007/s00466-014-1107-0.

[2] Charbel Farhat and Michael Lesoinne. Two efficient staggered algorithms for the serial and parallel solution of three-dimensional nonlinear transient aeroelastic problems. *Computer methods in applied mechanics and engineering*, 182(3-4):499–515, 2000.

[3] Carlos A Felippa, KC Park, and Charbel Farhat. Partitioned analysis of coupled mechanical systems. *Computer methods in applied mechanics and engineering*, 190 (24-25):3247–3270, 2001.

[4] Gene Hou, Jin Wang, and Anita Layton. Numerical methods for fluid-structure interaction—a review. *Communications in Computational Physics*, 12(2):337–377, 2012.

[5] Björn Hübner, Elmar Walhorn, and Dieter Dinkler. A monolithic approach to fluid–structure interaction using space–time finite elements. *Computer methods in applied mechanics and engineering*, 193(23-26):2087–2104, 2004.

[6] C Michler, SJ Hulshoff, EH Van Brummelen, and René De Borst. A monolithic approach to fluid–structure interaction. *Computers & fluids*, 33(5-6):839–848, 2004.

[7] Manuel Münsch. Entwicklung und anwendung eines semi-impliziten kopplungsverfahrens zur numerischen berechnung der fluid-struktur-wechselwirkung in turbulenten strömungen mittels large-eddy simulationen. 2015.

[8] Serge Piperno, Charbel Farhat, and Bernard Larrouturou. Partitioned procedures for the transient solution of coupled aroelastic problems part i: Model problem, theory and two-dimensional application. *Computer methods in applied mechanics and engineering*, 124(1-2):79–112, 1995.

[9] Thomas Richter. A monolithic multigrid solver for 3d fluid-structure interaction problems. 2010.

[10] EH Van Brummelen, SJ Hulshoff, and R De Borst. Energy conservation under incompatibility for fluid–structure interaction problems. *Computer Methods in Applied Mechanics and Engineering*, 192(25):2727–2748, 2003.