

Contents

1	Introduction	3
1.1	Outline	4
2	Medical Background	5
2.1	Anatomy of the central nervous system	5
2.2	The Spinal Cord	5
2.3	The Chiari malformation	6
2.3.1	Syringomyelia	6
2.3.2	Theories	7
3	Mathematical background	9
3.1	Fluid flow	10
3.1.1	The Divergence Theorem	10
3.1.2	Reynolds Transport Theorem	10
3.1.3	Conservation of mass and momentum	12
3.1.4	Incompressible Newtonian fluids	13
3.2	Navier-Stokes equations for incompressible flow	14
3.2.1	Boundary conditions	15
3.3	Linear Elasticity	17
3.4	Linear Poroelasticity	17
3.4.1	Fluid flow through a porous media	18
3.4.2	Biot's equations	18
3.5	Descriptions of Motion	20
3.5.1	Lagrangian and Eulerian descriptions of motion	20
3.5.2	The Arbitrary Eulerian Lagrangian description	21
3.6	Balance equations in the ALE framework	25
3.6.1	Mesh updating	25
3.7	Fluid Structure Interaction	25
3.8	Coupling Fluid Flow with Poroelasticity	26
4	Numerical Methods	29
4.1	The Finite Element Method	29
4.1.1	Variational formulation	29
4.1.2	Finite elements	31
4.2	The FEniCS software	32
4.3	A Discontinuous Galerkin method	34
4.3.1	Stokes flow	35
4.4	Womersley Flow	36

4.4.1	A penalty method for the boundary conditions	39
5	Numerical methods for FSI	43
5.1	A bencmark FSI-problem	43
5.1.1	Domain, Initial- and boundary conditions	44
5.2	Fluid Structure Interaction using the Finite Element method	45
5.2.1	Temporal discretization	45
5.2.2	Spatial discretization	45
5.2.3	A discussion on functionspaces	46
5.2.4	Treatment of boundary conditions	48
5.2.5	FSI in FEniCS	49
5.3	Benchmark Results	54
5.3.1	CFD tests	54
5.3.2	CSM tests	54
5.3.3	FSI tests	57
6	Numerical Methods for the Biot Problem	61
6.1	Weak form of the equations	62
7	Material parameters	63
8	Simulating interaction between CSF and the Spinal Cord	65
8.1	Overview of previous studies	65
8.1.1	Pressure measurements in patients with Chiari I	66
8.1.2	Applicability of medical measurements	67
8.2	CSF velocities in syringomyelia	68
8.3	A note on interface conditions	71
8.4	Results, elastic cord	71
8.4.1	No syrinx	72
8.4.2	1mm Syrinx	73
8.4.3	3mm Syrinx	73
8.5	Results, poroelastic cord	75
8.6	The effect of asymmetric pressure	75
8.7	Discussion	75
8.8	Limitations	76

Chapter 1

Introduction

The Cerebrospinal Fluid (CSF) surrounds the brain and acts as a protection to the brain inside the skull. As a result of the cardiac cycle, the CSF will flow up and down the subarachnoid space (SAS) surrounding the spinal cord. The Chiari malformation is a downwards displacement of a part of the brain known as the cerebellar tonsils that partially blocks CSF flow entering and leaving the SAS. This malformation is associated with syringomyelia, which is the presence of a fluid filled cavity within the spinal cord tissue. Treatment may include decompression surgery to remove parts of the bones of the skull to relieve pressure. Studies (e.g. Paul et al. (1983) [1], Lorenzo et al. (1995) [2], Guo et al. (2007)[3]) have shown that in many cases the syrinx gradually vanishes after surgery. The underlying mechanisms behind neither the formation nor the vanishing of the syrinx are not yet fully understood.

In vivo measurements by Quigley et al. (2004) [4] and Haughton et al. (2003) [5] have shown that the Chiari malformation is associated with abnormal CSF flow. Many researchers have thus suggested computational fluid dynamics (CFD) as a tool to give useful insight, as experiments are very difficult and expensive. These models have predicted abnormal CSF flow due to tonsillar herniation (Linge et al. (2011) [6]) as well as normalization of flow patterns modeling cases of post-operative craniocervical decompression surgery. (Linge et al. (2014) [7]) *In vitro* fluid-structure interaction (FSI) models by Martin et al. [8] have suggested that the presence and location of the syrinx have a critical impact on the pressure environment. Bertram et al. [9] have proposed in a series of *in silico* studies using FSI, that the so called 'slosh' mechanics may not generate sufficient force to lengthen a syrinx.

Kylstad (2014) [10] simulated the viscoelastic response of the spinal cord from applied pressure and compared displacement patterns between elastic and poroelastic models. Displacement patterns were shown to be similar but differ in magnitude depending on the material parameter setting. In the most extreme cases, displacement magnitudes were approximately 15 times larger for viscoelasticity, and a lag of 10 ms were observed compared to elastic models. These are both effects that can be triggered by altering parameters for the elastic spinal cord model. Støverud et al. (2015) [11] used poroelastic and viscoelastic models to describe the spinal cord, but to the authors knowledge poroelastic flow in the

spinal cord have not yet been coupled to the fluid flow in the SAS.
(Gabriela?)

The goal of the study is to model CSF flow around the cord and inside the syrinx by coupling computational models of fluid flow in the SAS with elastic and poroelastic models for the spinal cord. Changes in pressure and velocity distributions in the SAS as well as cord displacements, due to the presence of a syrinx will also be investigated.

1.1 Outline

The field of biomechanics requires multidisciplinary knowledge within medicine, mathematics, mechanics and numerical modeling. The authors main field of study as well as the main focus in this thesis lie in the last two.

In chapter 2, necessary background information to understand the physical problem from the physiological point of view is presented. The mathematical description of the physical problem based on laws of classical mechanics is given in chapter 3. The ALE-formulation, a necessary abstraction from the physical problem is also presented in this chapter. Chapter 4 gives an introduction to the FEniCS software including a few examples validating the CFD-solver used in the thesis. In chapter 5, the implementation of a FSI model is described and validated by comparing results to a benchmark problem. Implementation of the poroelastic model is given in chapter 6, and has been devoted a chapter itself due to differences in notation. Chapter 7 gives a justification of the material parameters used in simulations of the SAS–spinal cord–syrinx system presented in chapter 8. In chapter 9, discussion, limitations, summary and possible future work is given.

Chapter 2

Medical Background

In this chapter, a short description of the problem setting from a medical point of view is given. If not specified, the information given below is based on the textbook "Human Anatomy and Physiology", by Van Wynsberghe, Noback, and Carola [12]

2.1 Anatomy of the central nervous system

The human nervous system consists of peripheral nervous system, and the central nervous system (CNS). The former consists of spinal and cranial nerves and sensory receptor organs while the latter consists of two parts: the brain and the spinal cord. The CNS receives and processes information from all parts of the body. Consequently, studies on the CNS are crucial for our understanding of the human anatomy.

2.2 The Spinal Cord

The spinal cord carries information between the body and the brain and is divided into four or five regions from top to bottom: Cervical (C), thoracic (T), lumbar (L) and sacral (S) in addition to the coccygeal part at the very bottom. The upper end of the spinal cord is continuous with the lowermost part of the brain, and at the lower part tapers at the filum terminale, which attaches to the coccyx, known as the tailbone. Along the cord, there are 31 pairs of nerves exiting the cord: 8 in the cervical region, 12 in the thoracic region, 5 in the lumbar region, 5 in the sacral region and one in the coccygeal region. The first four segments of the cord are given names C1-C8, T1-T12, L1-L5, S1-S5. The tissue within the spinal cord consists of nervous tissue in form of white and grey matter which differs in both structure and function. In the center of the spinal cord, lies a tiny central spinal canal where CSF can flow. This channel closes off with age. Surrounding the central canal lies the gray matter in an H-shape, similar to a butterfly. The rest consists of white matter, and the ratio between white and gray matter differs along the spinal cord.

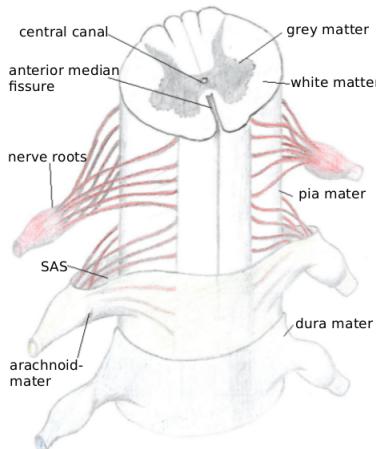


Figure 2.1: Schematic figure of the spinal cord. The pia mater surrounds the spinal cord, and between the pia mater and dura mater lies the subarachnoid space where CSF flows

There are three layers covering the brain and spinal cord known as meninges. The innermost layer surrounds the spinal cord and is known as the *pia mater*. The pia contains blood vessels that nourish the spinal cord. The middle layer, the *arachnoid* runs caudally extending almost all the way down the spinal cord. At the S2 vertebral level, the arachnoid joins the filum terminale. The outermost layer of the meninges protecting the spinal cord is known as the *dura mater* and is a though fibrous membrane.

2.3 The Chiari malformation

The Chiari malformation, also known as Arnold-Chiari Malformation is a neurological condition where a displacement of a part of the brain, the cerebellum, or more presice the cerebellar tonsils, down through the foramen magnum occur. The condition is classified into four stages I-IV where IV is the most severe. In a Chiari patient, the cerebellar tonsils obstructs the CSF flow (see figure 2.2) and even Chiari I patients have shown to have greater CSF velocities and a more complex flow pattern than healty subjects. [4] These patients could also experience severe headache, dizziness, tinnitus and muscle weakness. As the cerebellum is part of the brain that controls balance, loss of coordination have also been reported.

2.3.1 Syringomelia

In some cases, Chiari patients develops a fluid cavity, known as a syrinx within the spinal cord. Some of the symptoms are similar to the Chiari patients in

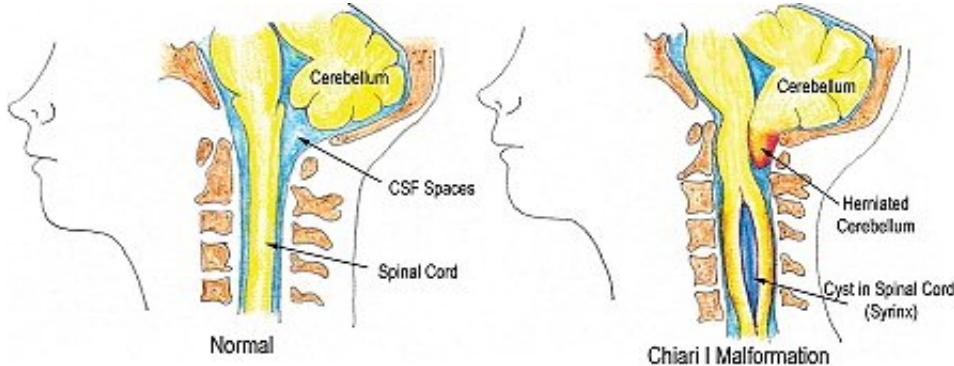


Figure 2.2: The Chiari malformation. Healthy subject to the left and a downward displacement into the CSF space known as the foramen magnum of the cerebellar tonsils on the right

general, but include muscle and back pain, weakness, numbness and inabilities to feel temperature changes. Many theories on the pathogenesis of syringomelia have been proposed, but the details are not yet fully understood. In patients diagnosed with Chiari I, about 2/3 develops syrinxes within the spinal cord tissue. Thus, not all Chiari patients have syringomelia and in addition patients with syringomyelia does not necessarily have the Chiari malformation.

Thompson et al. (2015) [13] have in a recent study suggested the anatomy of the cervical (upper part) spinal canal plays a role in the formation of syrinxes. In healthy subjects the spinal canal tapers from the The study showed that Chiari patients with syringomelia had positive *C4-C7*-tapering (expanding caudally) of the cervical spinal canal, but no significant difference were found on *C1-C4*-tapering.

Even though researchers do not seem to agree on the underlying mechanics leading to Syringomyelia, most seem to agree on the fact that altered CSF dynamics is associated with the formation of the syrinx.

2.3.2 Theories

[14]

Chapter 3

Mathematical background

The flow of CSF around the spinal cord requires equations for fluid flow to be coupled with equations for elasticity or in the optimal case, poroelasticity. The underlying concepts of these kinds of problems were originally developed somewhat independently within petroleum engineering, geomechanics and hydrogeology. The equations will first be presented separately. Later in the chapter, coupling conditions will be discussed. Several quantities will be discussed and as far as possible, we try to use a consistent notation for each quantity throughout the study.

This chapter intends to give a short description of the mathematical theory behind modeling CSF. The equations are first introduced by assuming a fixed set of coordinates. Later, the two fundamental descriptions of motion are discussed.

In providing the necessary mathematical background it is convenient to give a overview of the notation used. If not specifically specified otherwise we use:

v - Velocity of the material. In the fluid, **v** represents fluid velocity, and in the solid **v** denotes the velocity of the solid.

U - Total displacement of the solid. In the fluid, this quantity represents the total mesh displacement.

p - Pressure in the fluid. In the case of elasticity, the same pressure does not exist in the solid, and in the case of poroelasticity *p* represent the pore pressure inside the fluid/solid mixture.

w - Domain velocity, (i.e, all material points within the domain moves with velocity **w**). In the case of a pure elastic medium, the solid domain changes with the velocity of the solid, **v**, so in the solid we have **v** = **w** which will not neccessarily hold in the fluid.

Unit vectors in the cartesian coordinate system are denoted **i** = (1, 0, 0), **j** = (0, 1, 0) and **k** = (0, 0, 1). For summation convention the common choice **i_i**, **i_j**, **i_k** are used respectively.

The index notation to denote components of vectors and tensors are used together with Einstein summation convention $\mathbf{v}_i \mathbf{i}_i = v_1 \mathbf{i}_1 + v_2 \mathbf{i}_2 + v_3 \mathbf{i}_3$ for a vector $\mathbf{v} = (v_1, v_2, v_3)$ are used occasionally. A sum is taken over a repeated index.

It should be noted that the definition of ∇ differs in the literature. We define ∇ as a tensor, $\nabla = \mathbf{i}_i \frac{\partial}{\partial x_i}$, and thus $\nabla \mathbf{v} = \frac{\partial v_i}{\partial x_j} \mathbf{i}_j \mathbf{i}_i$

Subscripts f and s are used to denote fluid and solid quantities, respectively

3.1 Fluid flow

The most fundamental equations in fluid flow are conservation laws. These equations are based on classical mechanics and states conservation of mass, momentum and energy. In the literature, these are often referred to as balance equations.

3.1.1 The Divergence Theorem

The divergence theorem is result that relates the flow of a *vector field* through a closed surface to the divergence of the vector field inside the surface. The theorem is usually credited to Green or Gauss, but other mathematicians also contributed (see e.g. [15] for a brief history). The divergence theorem states that for a vector field \mathbf{F} in a region V_0 bounded by a closed surface S_0

$$\int_{V_0} \nabla \cdot \mathbf{F} dV = \int_{S_0} \mathbf{F} \cdot \mathbf{n} dS \quad (3.1)$$

where \mathbf{n} is the outward unit normal on S_0 . For a second rank *tensor*, $\mathbf{T} = T_{ij} \mathbf{i}_i \mathbf{i}_j$ for $i, j = 1, 2, 3$, the divergence theorem becomes a vector equation, and will differ slightly with the tensor notation used in this thesis. We have

$$\nabla \cdot \mathbf{T} = \mathbf{i}_i \frac{\partial}{\partial x_i} \cdot T_{kj} \mathbf{i}_k \mathbf{i}_j = \frac{\partial T_{ij}}{\partial x_i} \mathbf{i}_j$$

To ensure the divergence theorem to hold for each direction \mathbf{i}_j , the volume integral over this expression must equal the surface integral of

$$T_{ij} n_i \mathbf{i}_j = n_i \mathbf{i}_i \cdot T_{jk} \mathbf{i}_j \mathbf{i}_k = \mathbf{n} \cdot \mathbf{T}$$

And therefore, the divergence theorem for a second-rank tensor becomes

$$\int_{V_0} \nabla \cdot \mathbf{T} dV = \int_{S_0} \mathbf{n} \cdot \mathbf{T} dS \quad (3.2)$$

For a symmetric tensor, $\mathbf{n} \cdot \mathbf{T} = \mathbf{T} \cdot \mathbf{n}$ and we can use the form (3.1). However, when dealing with non-symmetric tensors the form (3.2) must be used.

3.1.2 Reynolds Transport Theorem

The famous engineer and scientist Osbourne Reynold stated the general conservation law the following way [16]:

Any change whatsoever in the quantity of any entity within a closed surface can only be effected in one or other of two distinct ways:

1. it may be effected by the production or destruction of the entity within the surface, or
2. by the passage of the entity across the surface.

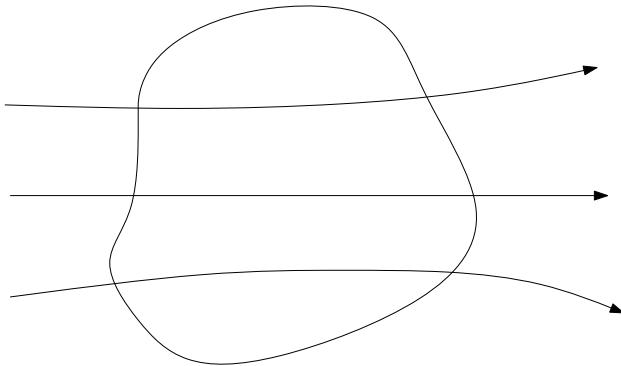


Figure 3.1: Fixed control volume with flow as indicated by streamlines

It should be noted that the transport theorem can be approached in two different ways. One for a fixed set of spatial coordinates, a fixed control volume, where fluid can enter and exit the boundaries of the defined body. The other approach has a control volume consisting of the same material particles at all times. Therefore the body has to follow the flow, and no fluid will cross the boundary. In this approach, one has to take into account the movement of the boundary of the body and the fact that the body can change its volume. More on descriptions of motion is described in section 3.5

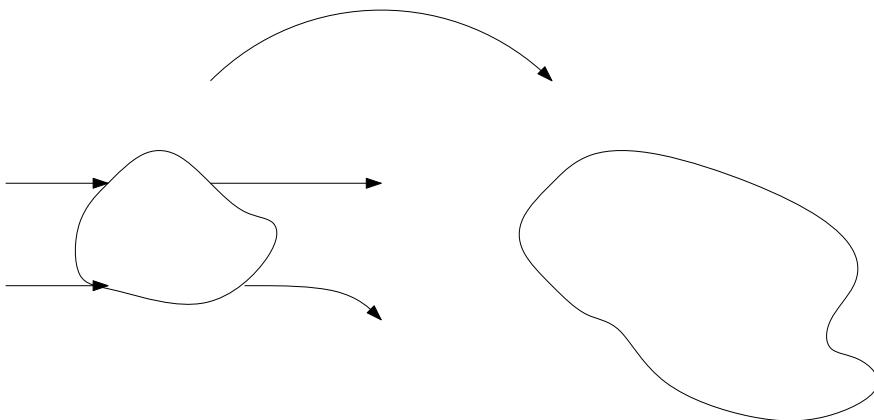


Figure 3.2: A moving control volume, consisting of the same fluid particles at time t (left) and time $t + \Delta t$ (right)

Now, consider a fixed control volume, V_0 and some fluid property $Q(\mathbf{x}, t)$. The rate of change of Q within the control volume can be written

$$\frac{d}{dt} \int_{V_0} Q(\mathbf{x}, t) dV$$

The net change of Q must be equal the rate of change in Q within the control volume plus the net rate of mass flow out of the volume. In other words:

$$\frac{d}{dt} \int_{V_0} Q(\mathbf{x}, t) dV = \int_{V_0} \frac{\partial Q(\mathbf{x}, t)}{\partial t} dV + \int_{S_0} Q(\mathbf{x}, t) \mathbf{v} \cdot \mathbf{n} dS$$

Here, \mathbf{v} denotes fluid velocity, and \mathbf{n} denotes the outward pointing unit-normal, i.e \mathbf{n} points *out* of the fluid. This equation is known as the Reynold's transport theorem. The right hand side could be rewritten by using Gauss' theorem on the last term.

$$\frac{d}{dt} \int_{V_0} Q(\mathbf{x}, t) dV = \int_{V_0} \left[\frac{\partial Q(\mathbf{x}, t)}{\partial t} + \nabla \cdot (Q(\mathbf{x}, t) \mathbf{v}) \right] dV \quad (3.3)$$

3.1.3 Conservation of mass and momentum

Choose $Q(\mathbf{x}, t) = \rho$, where ρ is fluid density. Conservation of mass means that

$$\frac{d}{dt} \int_{V_0} \rho dV = 0$$

And by using the transport theorem (3.3)

$$\int_{V_0} \left[\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) \right] dV = 0$$

This should hold for any volume V_0 , hence the integrand has to be zero. If there existed a point where the integrand were not zero, the control volume could be an arbitrary small enclosed sphere around this point, and the volume integral would not be zero. Therefore,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \quad (3.4)$$

Equation (3.4) is known as the continuity equation and states conservation of mass.

To derive a simliar property for the momentum, Newtons second law of motion can be used. The net change of momentum must be equal to the applied forces to the system. The forces can be divided into volume forces, acting on the entire control volume, and forces acting only on the control surface. The forces acting on the surface can be written $\sigma_f \cdot \mathbf{n}$, where $\sigma_f = \sigma_f(\mathbf{v}, p)$ is the (symmetric) tensor denoting the total stress.

This can be written

$$\frac{d}{dt} \int_{V_0} \rho \mathbf{v}(\mathbf{x}, t) dV = \int_{\partial V_0} \sigma_f \cdot \mathbf{n} dS + \int_{V_0} \mathbf{F}_v dV$$

By using the Transport Theorem on the left hand side and with Gauss' theorem on the right hand side we end up with

$$\int_{V_0} \left[\frac{\partial \rho \mathbf{v}}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) - \nabla \cdot \sigma_f - \mathbf{F}_v \right] dV = 0$$

With the same argument as before the integrand has to be zero, and with some rearrangement:

$$\frac{\partial \rho \mathbf{v}}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = \nabla \cdot \sigma_f + \mathbf{F}_v \quad (3.5)$$

The left hand side can be rewritten to

$$\rho \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \frac{\partial \rho}{\partial t} + \mathbf{v} \mathbf{v} \cdot \nabla \rho + \rho \mathbf{v} \cdot \nabla \mathbf{v} + \mathbf{v} \rho \cdot \nabla \mathbf{v} = \rho \left(\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} \right) + \left(\frac{\partial \rho}{\partial t} + \nabla \cdot (\mathbf{v} \rho) \right)$$

And the last term is zero due to mass conservation. Equation (3.5) can then be written

$$\rho \left(\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} \right) = \nabla \cdot \sigma_f + \mathbf{F}_v \quad (3.6)$$

The stress tensor, σ_f , depends on fluid properties and will be defined in the next subsection. Equation (3.6) is known as the momentum equation as it states conservation of momentum.

3.1.4 Incompressible Newtonian fluids

In this text we will only consider incompressible fluid flow for a Newtonian fluid. The information given below is based on explanations by White, [17, pp. 65-66] and Gjevik [18]

The assumption of a Newtonian fluid requires the viscous stresses to be linear functions of the components of the strain-rate tensor, denoted by ϵ . These assumptions were first made by Stokes in 1845. Stokes' assumptions have later proven to be quite accurate for all gases and most common fluids. Stokes' three postulates regarding the deformation laws are:

1. The fluid is continuous, and its stress tensor, $\sigma_{f_{ij}}$ is at most a linear function of the strain rates, ϵ_{ij}
2. The fluid is isotropic, i.e., its properties are independent of direction, and therefore the deformation law is independent of the coordinate axes in which it is expressed.
3. When the strain rates are zero, the deformation law must reduce to the hydrostatic pressure condition, $\sigma_{f_{ij}} = -p\delta_{ij}$, where δ_{ij} is the Kroenecker delta function.

From the first and third condition the following assumption can be made

$$\sigma_{f_{ij}} = -p\delta_{ij} + M_{ijkl}\epsilon_{kl} \quad (3.7)$$

As done by Gjevik, listing each component, it can be shown that symmetry of σ_f and ϵ also requires symmetry of M . This assumption reduces the number of coefficients in equation (3.7) from 36 to 21. If Stokes' second condition is also taken into account and the fluid properties are identical in each direction, the number of coefficients are further reduced to 2. These simplifications allow us to denote the stress tensor the following way:

$$\sigma_{f_{ij}} = -p\delta_{ij} + 2\mu\epsilon_{ij} + \lambda\nabla \cdot \mathbf{v} \quad (3.8)$$

where $\epsilon_{i,j} = \frac{1}{2}(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i})$, p is the fluid pressure and μ and λ are known as Lame's constants. In the present study we only consider incompressible flow where ρ is constant. From the continuity equation (3.4), this implies $\nabla \cdot \mathbf{v} = 0$ and the last term in equation (3.8) vanishes. Furthermore,

$$\nabla \cdot 2\mu\epsilon = \mu \frac{\partial}{\partial x_j}(\frac{\partial v_j}{\partial x_i} + \frac{\partial v_i}{\partial x_j})\mathbf{i}_i = \mu(\frac{\partial}{\partial x_i}\frac{\partial v_j}{\partial x_j} + \frac{\partial v_i}{\partial x_j}\frac{\partial v_j}{\partial x_i})\mathbf{i}_i = \mu \frac{\partial v_i}{\partial x_j}\frac{\partial v_j}{\partial x_i}\mathbf{i}_i = \mu \nabla^2 \mathbf{v}$$

Which simplifies the representation of $\nabla \cdot \sigma_f$ in (3.6) for an incompressible fluid

3.2 Navier-Stokes equations for incompressible flow

The system of equations (3.4),(3.6) are commonly referred to as the Navier-Stokes equations written in *divergence form*, where the Cauchy stress tensor, σ is explicitly included in the momentum equation and contributes to the momentum through its divergence. However, in the case of Newtonian incompressible fluids, the simplifications described in the previous section allows us to write the system of equations in *laplace form*

$$\rho(\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla)\mathbf{v}) = -\nabla p + \mu \nabla^2 \mathbf{v} + \mathbf{F}_v$$

$$\nabla \cdot \mathbf{v} = 0$$

The parameters ρ and μ describe fluid density and dynamic viscosity. Often, the momentum equation is written in terms of the kinematic viscosity $\nu = \frac{\mu}{\rho}$ by dividing the momentum equation with ρ .

It should be noted that the two formulations of the momuentum equation are equivalent in their original form. In textbooks, see e.g. [19, 17, 20], the laplace form is usually the form first presented as the Navier-Stokes equations probably because it is the simplest form explicitly including the two unknowns \mathbf{v} and p .

The Navier-Stokes equations are coupled and non-linear and can generally not be solved analytically. However, numeruous analytical solutions have been carried out for different specific problems, and a good overview is given by White [17, pp. 97-164]. (Further references are also given in this textbook for the interested reader) These problems are often very simple and idealized. Hence, numerical solutions are a necessity to obtain useful solutions to real-life problems. Such metods will be discussed in chapter 4.

3.2.1 Boundary conditions

Before the equations can be solved, appropriate boundary conditions need to be imposed on all boundaries of the domain. For a specific fluid occupying a domain, the treatment of boundary conditions is what distinguishes different flow patterns as the governing equations inside the domain stay exactly the same.

A fluid generally moves around between solid boundaries, $\partial\Omega_D$ and boundaries known as traction boundaries $\partial\Omega_N$.

On $\partial\Omega_D$ e.g. the interface between a fluid and a solid wall, the fluid velocity must equal the wall velocity in all directions, often known as the no-slip boundary condition. For instance at a rigid wall, the boundary condition will be $\mathbf{v} = 0$ on $\partial\Omega_D$, or in general

$$\mathbf{v} = \mathbf{v}_0 \text{ on } \partial\Omega_D$$

On $\partial\Omega_N$ external forces on the system must be imposed. At surfaces where arbitrary forces \mathbf{F} are acting, the fluid stress on the surface must equal these external forces. This can be written

$$\sigma \cdot \mathbf{n} = \mathbf{F} \text{ on } \partial\Omega_N$$

Where \mathbf{n} is the outward normal unit vector.

This is also the case of a free surface, except the shear forces are negligible and that only external pressures are applied. This yields

$$\sigma \cdot \mathbf{n} = -p_0 \mathbf{n} \text{ on } \partial\Omega_N$$

where p_0 is a known external pressure, for instance atmospheric pressure at the ocean surface.

A third option is the so called pseudo-traction boundary condition, where we set

$$\mu \frac{\partial \mathbf{v}}{\partial n} - p \mathbf{n} = -p_0 \mathbf{n} \text{ on } \partial\Omega_N \quad (3.9)$$

where p_0 is some prescribed pressure. With the tensor notation of ∇ , the normal derivative is defined as $\frac{\partial \mathbf{v}}{\partial n} = \mathbf{n} \cdot \nabla \mathbf{v}$. This boundary condition is often associated with the Laplace form of the Navier-Stokes equation as it naturally appears on the boundary when integrating the weak form of the Laplace term by parts.

It should also be noted that the physical implications between the pseudo-traction condition is different from the external force boundary condition $\sigma \cdot \mathbf{n} = \mathbf{F}$. Let's assume we have a two-dimensional horizontal channel, and take a moment to examine the interpretation, or physical implications, of the pseudo-traction condition (3.9) on the outlet ($\mathbf{n} = (1, 0)$)

$$\mu \frac{\partial \mathbf{v}}{\partial n} - p \mathbf{n} = -p_0 \mathbf{n}.$$

With unit-vectors \mathbf{i} and \mathbf{j} in the x- and y-direction and the velocity vector $\mathbf{v} = (v_1, v_2)$, the two components can be written

$$\begin{aligned}\mu \frac{\partial v_1}{\partial x} - p &= -p_0, \\ \mu \frac{\partial v_2}{\partial x} &= 0.\end{aligned}$$

The second condition can be interpreted as having the vertical component, v_2 equal just outside and just inside the domain. This should mimic a continuation of the channel under the assumption that $v_2 = 0$ inside the channel which is valid due to mass conservation.

A condition on $\sigma \cdot \mathbf{n}$ on the boundary is a more general approach for setting external forces on the boundary. The physical implications of no external forces on the outlet in the previous example could be compared to a garden hose, where water can exit in all directions and creep around the corners of the outlet.

$$\mu \frac{\partial \mathbf{v}}{\partial \mathbf{n}} - p \mathbf{n} = 0$$

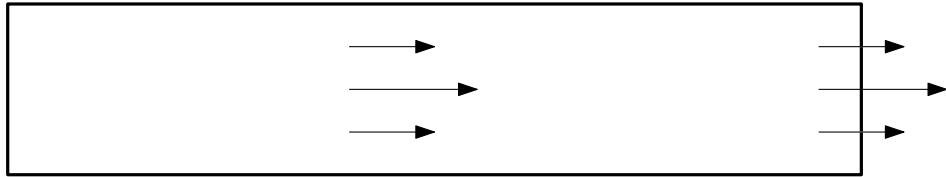


Figure 3.3: The pseudo traction boundary condition implies a continuation of the channel

$$\sigma \cdot \mathbf{n} = 0$$

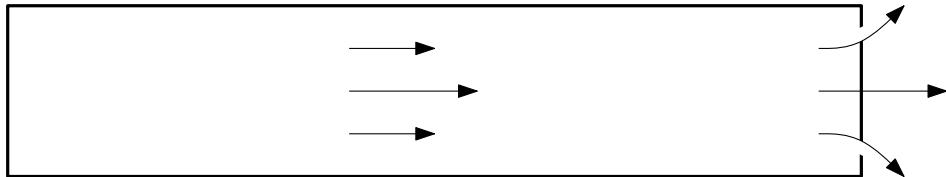


Figure 3.4: No external forces implies an open end of the channel, and fluid can escape in all directions

3.3 Linear Elasticity

The equations describing elasticity is derived by using Reynolds Transport theorem on a moving domain, Ω^t , consisting of the same particles at all times. For conservation of momentum, the change in momentum must equal the applied forces to the system as well as body forces.

$$\frac{d}{dt} \int_{\Omega^t} \rho \mathbf{v} dV = \int_{\Omega^t} \rho \frac{\partial \mathbf{v}}{\partial t} dV + \int_{\partial \Omega^t} \rho \mathbf{v} \mathbf{v} \cdot \mathbf{n} dS = \int_{\partial \Omega^t} \sigma_s \cdot \mathbf{n} dS + \int_{\Omega^t} \mathbf{F}_v dV$$

Where σ_s is the stress tensor describing the elastic material. By applying Gauss' theorem again we end up with the general elasticity equation in a moving domain.

$$\rho \frac{\partial \mathbf{v}}{\partial t} + \rho (\mathbf{v} \cdot \nabla) \mathbf{v} = \nabla \cdot \sigma_s + \mathbf{F}_v \quad (3.10)$$

Linear elasticity is an approximation used for small deformations for elastic solids. As a rule of thumb the approximation of linear elasticity is usually valid for deformations up to 10% relative to the solid. The stress tensor for a linear elastic medium is very similar to (3.8) describing an incompressible Newtonian fluid except there is no fluid pressure, and the stress is related to the total displacement \mathbf{U} rather than the velocity \mathbf{v} . The stress tensor for such a material reads: $\sigma_s = 2\mu\epsilon(\mathbf{U}) + \lambda \text{tr}(\epsilon(\mathbf{U}))$, where ϵ is defined exactly as the strain rate tensor for a Newtonian fluid. Usually, within the framework of linear elasticity, the convective term is also regarded as small and thus neglected. It is then common to write the equation only involving one unknown, \mathbf{U} .

We choose to keep the equation (3.10) for simulations in this thesis. The reason for choosing the solid velocity \mathbf{v} as the unknown will be discussed in chapter 5. Also, by using this form, the nonlinear term add no complexity to the coupled system we want to solve later, as the momentum equation in the fluid will have a similar term. Thus, our linear elasticity approximation lies in the inexact description of σ_s which in general will consist of nonlinear terms depending on the elasticity model.

3.4 Linear Poroelasticity

In this section, the equations describing fluid flowing through an elastic, porous medium is presented. For a more detailed discussion, derivation and history within the field we refer to Wang [21] on Linear Poroelasticity. To keep the mathematics as similar to the fluid case as possible, we use μ and λ instead of the Poisson ratio. The equations describing linear poroelasticity are often described as Biot's equations.

3.4.1 Fluid flow through a porous media

A porous medium is a material with pores in which fluid can flow. The principles of modeling porous flow consist of macroscopic averaging over the pores. The material part is often known as the skeleton, matrix or frame and in general all the pores will have different size and shape. In this section the skeleton is assumed rigid. The nature of the material defines whether we will be able to fully solve the problem with no-slip conditions on all skeleton parts, or if some kind of volumetric averaging can be done. If the observer is interested in velocity variations on the scales of the pores, conventional fluid dynamics must be used. When there are many pores and channels, the complexity of the problem makes the full Navier-Stokes system difficult to solve. In these cases, macroscopic volume averaging is usually done, where the effects of the skeleton is modeled by introducing parameters constant over the material considered, such as permeability and conductivity.

These kinds of simplifications results in the famous Darcy's law (see e.g. Nield and Bejan [22]), here generalized in three dimensions

$$\mathbf{q} = -\frac{1}{\mu} \mathbf{K} \cdot \nabla p. \quad (3.11)$$

Here \mathbf{K} is the *permeability* tensor, and μ is the dynamic viscosity of the fluid. If the medium considered is isotropic, the permeability is a scalar value and equation (3.11) can be expressed as

$$\mathbf{q} = -\frac{K}{\mu} \nabla p$$

Sometimes the permeability is given through the *hydraulic conductivity*, κ , where $\kappa = \frac{K}{\mu_f}$ relates the two parameters for a fully saturated porous medium. It should be noted that the velocity \mathbf{q} known as the Darcy velocity (or Darcy flux) represents the average flux over a representative elementary volume (r.e.v.), and thus the fluid velocity experienced by a particle in the pores will be $\mathbf{v}_p = \frac{\mathbf{q}}{\phi}$ where ϕ is the *porosity* and describes the ratio of the pore volume versus the total volume. i.e. a high porosity indicates a large volume of pores compared to the skeleton and in general less obstruction of fluid.

3.4.2 Biot's equations

As in the previous section, we let the domain consist of a skeleton with fluid filled pores. The extension for the Biot problem is that the skeleton is now free to move as an elastic material. Incompressibility for the fluid and solid is assumed. We define the filtration velocity $\mathbf{q} = \phi(\mathbf{v}_p - \mathbf{v}_s)$ where \mathbf{v}_p is the fluid velocity in the pores and \mathbf{v}_s is the structural velocity. \mathbf{q} is thus regarded as the relative velocity of the fluid compared to the solid. The velocity of the skeleton \mathbf{v} , the filtration velocity \mathbf{q} , and the pore pressure p , can now be related by the

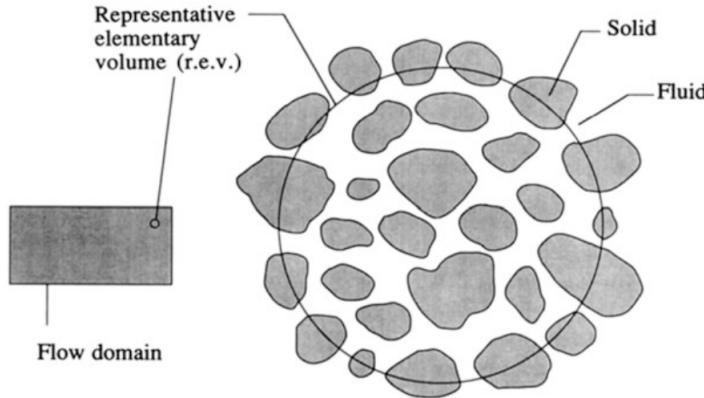


Figure 3.5: Representation of porous media where the averaging approach is used. From Nield and Bejan

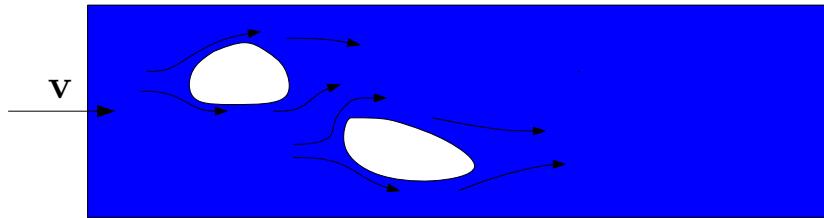


Figure 3.6: River flow around two islands representing the pores. The full Navier-Stokes system should be solved for this problem

following set of equations[23],[24],[25]

$$\rho_p \frac{d\mathbf{v}_s}{dt} + \rho_f \frac{d\mathbf{q}}{dt} + \nabla p - \nabla \cdot \sigma_s(\mathbf{U}) = \mathbf{f}_s \quad (3.12)$$

$$\rho_f \frac{d\mathbf{v}_s}{dt} + \rho_f \frac{d\mathbf{q}}{dt} \frac{1}{\phi} + K^{-1} \mathbf{q} + \nabla p = \mathbf{f}_d \quad (3.13)$$

$$\nabla \cdot (\mathbf{v}_s + \mathbf{q}) = 0 \quad (3.14)$$

These equations state conservation of momentum equation for the total force balance (3.12), conservation of momentum for the fluid phase only, (3.13) and the constraint of incompressibility (3.14). ρ_f is the density of the fluid in the pores, and $\rho_p = \rho_s(1 - \phi) + \rho_f\phi$ where ρ_s is the density of the skeleton.

In relation to Darcy's law without external forces, equation (3.13) describes an extension both in terms of the material derivative of \mathbf{q} , as well as extension into the poroelastic regime. The former is an extension to Darcy's law as proposed by Nield and Bejan, where originally we have used $\frac{1}{\phi}$ as the acceleration coefficient tensor. Biot's equations have been much used in applied geoscience and hydrogeology, where the time derivative $\frac{d\mathbf{q}}{dt}$ is small. This assumption is probably less valid for some applications within biomedical computing, however as the spinal cord has previously been believed to be impermeable, and also accord-

ing to previous results by Drøsda [26], we expect \mathbf{q} to be small and that the material derivative can be dropped. The term $K^{-1}\mathbf{q}$ should be kept as K is assumed to be comparable to \mathbf{q} in orders of magnitude.

3.5 Descriptions of Motion

The conservation equations for Newtonian fluids were derived from Reynolds' transport theorem by using a control volume fixed in space, while for elastic materials a moving control volume was used. In addition, we saw for instance that the stress tensor for elastic solids were linked to the total displacement, or deviation from the stress-free configuration, in the material. The stresses and velocity in the material will depend on the current deformation of the material with respect to the stress-free configuration.

To this end, it will be convenient to provide the reader with two classical descriptions of a continuum in motion.

3.5.1 Lagrangian and Eulerian descriptions of motion

We consider a domain $\Omega_{\mathbf{X}} \in \mathbb{R}^3$ consisting of material particles \mathbf{X} . The domain can undergo deformations, and the deformed domain, $\Omega_{\mathbf{x}}$, is the current configuration at time t .

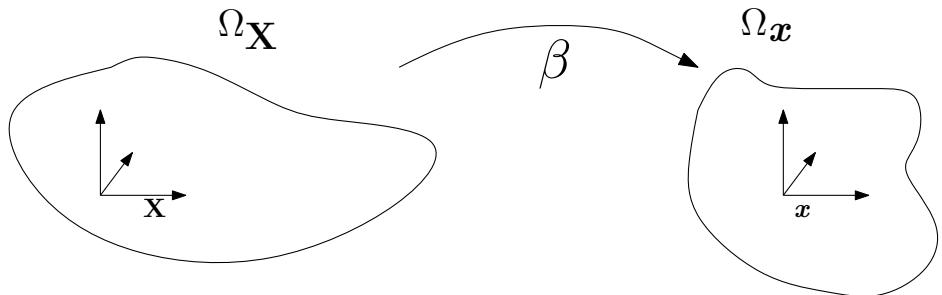


Figure 3.7: Lagrangian description of motion. The mapping β maps the reference coordinates to the spatial ones

We define the one-to-one mapping:

$$\begin{aligned}\beta : \Omega_{\mathbf{X}} \times [0, T] &\rightarrow \Omega_{\mathbf{x}} \times [0, T] \\ (\mathbf{X}, t) &\rightarrow \beta(\mathbf{X}, t) = (\mathbf{x}, t)\end{aligned}$$

Which takes any point \mathbf{X} in the reference configuration to a new position $\mathbf{x} = \beta(\mathbf{X}, t)$ at time t . As the mapping is one-to-one, it is also possible to keep track of the history of the motion by the inverse, β^{-1} . The time is measured with the same variable, t , in both domains. The gradient of β with respect to (\mathbf{X}, t) can be written in matrix form as:

$$\frac{\partial \beta}{\partial (\mathbf{X}, t)} = \begin{pmatrix} \frac{\partial \mathbf{x}}{\partial \mathbf{X}} & \mathbf{v} \\ 0^T & 1 \end{pmatrix}$$

where the material velocity

$$\mathbf{v}(\mathbf{X}, t) = \frac{\partial \mathbf{x}}{\partial t} \Big|_{\mathbf{X}} \quad (3.15)$$

is the temporal change in the spatial variable \mathbf{x} while holding \mathbf{X} fixed. 0^T denotes a null vector.

The *Lagrangian* description, where we follow a fixed set of material particles as suggested by the mapping β , is often used. In the Lagrangian description all quantities are expressed in terms of the reference configuration $\Omega_{\mathbf{X}}$ and time. In other words, even though the material is deformed, we can still compute displacements and particle velocities using the material coordinates \mathbf{X} . For instance, the displacement from the starting material configuration will be given as $\beta(\mathbf{X}, t) - \mathbf{X}$ and the velocity as given in equation (3.15).

Because the grid coincides with the material coordinates, there are no convective terms in the Lagrangian description. In the context of Reynold's Transport theorem, the Lagrangian approach coincides with a moving control volume consisting of the same material points at all time. When a material undergoes large deformations or for instance vortices or turbulence occur, the material velocity from the Lagrangian point of view becomes difficult to handle.

In fluid mechanics the *Eulerian* description is the most used, which means that fluid flows through a fixed region in space and in each point we can measure various properties or quantities such as velocity, pressure and temperature. The conservation equations in the Eulerian description are expressed in terms of the spatial coordinates \mathbf{x} and time, and are neither connected to a reference configuration nor the material coordinates. Compared to the Lagrangian approach, large material deformations is not a problem, as material can enter and leave the fixed domain. This movement of a material through a fixed region results in convective effects, and convection operators can often be problematic in computational fluid dynamics due to their non-symmetric nature.

3.5.2 The Arbitrary Eulerian Lagrangian description

To be able to couple the Lagrangian approach for the solid with the Eulerian approach for the fluid, we need some referential system, not attached to the material points neither totally fixed in space. This type of description is common in FSI analysis and is known as the *arbitrary Lagrangian-Eulerian* (ALE) description.

The following derivation is inspired by the the works on Arbitrary Lagrangian-Eulerian methods by Donea et. al in [27].

The need for an additional set of coordinates, an independent referential system with reference coordinates χ is introduced. This introduces two new mappings to relate all the different configurations as shown in fig.

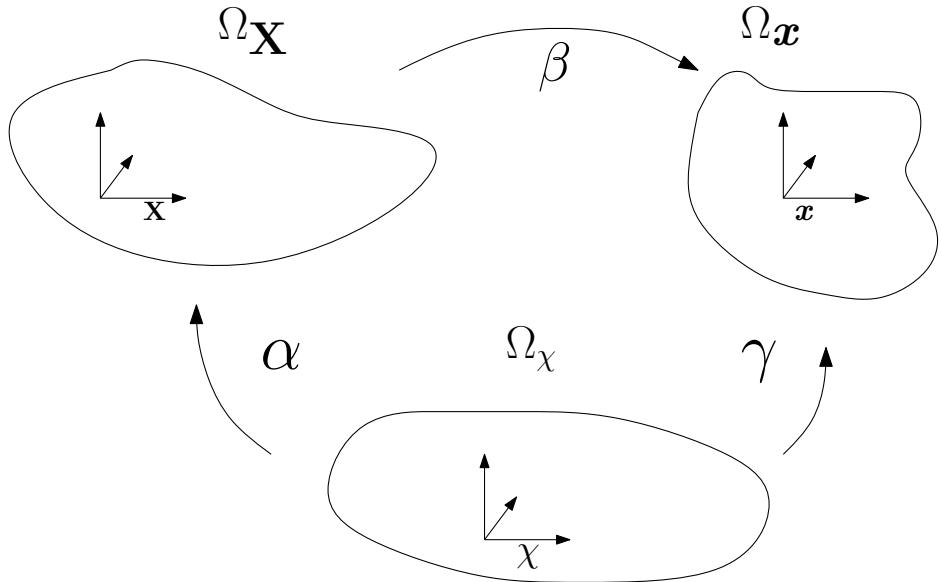


Figure 3.8: The three domains needed in the ALE formulation

The mappings are defined similarly to β as

$$\begin{aligned}\gamma : \Omega_\chi \times [0, T] &\rightarrow \Omega_x \times [0, T] \\ (\chi, t) &\rightarrow \gamma(\chi, t) = (x, t)\end{aligned}$$

and the gradient of γ

$$\frac{\partial \gamma}{\partial (\chi, t)} = \begin{pmatrix} \frac{\partial x}{\partial \chi} & \mathbf{w} \\ 0^T & 1 \end{pmatrix}$$

In addition,

$$\mathbf{w}(\chi, t) = \frac{\partial \mathbf{x}}{\partial t} \Big|_\chi$$

denotes the mesh velocity. Both the mesh and the material can move independently of the laboratory. More precise, relative to some referential point in space, the fluid moves with velocity \mathbf{v} and the domain moves with velocity \mathbf{w} .

To complete the relation between the different velocities, we define the inverse of α directly:

$$\begin{aligned}\alpha^{-1} : \Omega_\mathbf{x} \times [0, T] &\rightarrow \Omega_\chi \times [0, T] \\ (\mathbf{x}, t) &\rightarrow \alpha^{-1}(\mathbf{x}, t) = (\chi, t)\end{aligned}$$

The gradient is given as

$$\frac{\partial \alpha^{-1}}{\partial (\mathbf{x}, t)} = \begin{pmatrix} \frac{\partial \chi}{\partial \mathbf{x}} & \hat{\mathbf{v}} \\ 0^T & 1 \end{pmatrix}$$

Where the velocity

$$\hat{\mathbf{v}}(\mathbf{X}, t) = \frac{\partial \chi}{\partial t} \Big|_{\mathbf{x}} \quad (3.16)$$

denotes the temporal change in the referential system while holding the material particle \mathbf{X} fixed. Therefore the velocity $\hat{\mathbf{v}}$ can be interpreted as the particle velocity in the referential domain.

We use that $\beta = \gamma \circ \alpha^{-1} = \gamma(\alpha^{-1}(\mathbf{X}, t))$ and obtain a relation between the different velocities by differentiating β .

$$\begin{aligned} \frac{\partial \beta}{\partial(\mathbf{X}, t)}(\mathbf{X}, t) &= \frac{\partial \gamma}{\partial(\chi, t)}(\alpha^{-1}(\mathbf{X}, t)) \frac{\partial \alpha^{-1}}{\partial(\mathbf{X}, t)}(\mathbf{X}, t) \\ &= \frac{\partial \gamma}{\partial(\chi, t)}(\chi, t) \frac{\partial \alpha^{-1}}{\partial(\mathbf{X}, t)}(\mathbf{X}, t) \end{aligned}$$

In matrix form this equation is written:

$$\begin{pmatrix} \frac{\partial \mathbf{x}}{\partial \mathbf{X}} & \mathbf{v} \\ 0^T & 1 \end{pmatrix} = \begin{pmatrix} \frac{\partial \mathbf{x}}{\partial \chi} & \mathbf{w} \\ 0^T & 1 \end{pmatrix} \begin{pmatrix} \frac{\partial \chi}{\partial \mathbf{X}} & \hat{\mathbf{v}} \\ 0^T & 1 \end{pmatrix}$$

After block-multiplication of the right hand side we end up with an equation relating all the different velocities:

$$\mathbf{v} = \frac{\partial \mathbf{x}}{\partial \chi} \cdot \hat{\mathbf{v}} + \mathbf{w}$$

To this end, it is convenient to define the convective velocity

$$\mathbf{c} := \mathbf{v} - \mathbf{w} = \frac{\partial \mathbf{x}}{\partial \chi} \cdot \hat{\mathbf{v}}$$

which is the relative velocity between the material and the mesh.

To obtain relation between quantities to formulate the balance equations, we let a scalar quantity, Q be defined as $Q(\mathbf{x}, t)$, $Q^*(\chi, t)$ and $Q^{**}(\mathbf{X}, t)$ in the spatial, referential and material domains respectively.

To obtain a relation between the spatial description, Q , and material description Q^{**} we can utilize the previously described mapping β :

$$Q^{**}(\mathbf{X}, t) = Q(\beta(\mathbf{X}, t), t) = Q \circ \beta$$

The gradient of Q^{**} can then be computed as

$$\frac{\partial Q^{**}}{\partial(\mathbf{X}, t)}(\mathbf{X}, t) = \frac{\partial Q}{\partial(\mathbf{x}, t)}(\mathbf{x}, t) \frac{\partial \beta}{\partial(\mathbf{X}, t)}(\mathbf{X}, t) \quad (3.17)$$

$$\begin{pmatrix} \frac{\partial Q^{**}}{\partial \mathbf{X}} & \frac{\partial Q^{**}}{\partial t} \\ 0^T & 1 \end{pmatrix} = \begin{pmatrix} \frac{\partial Q}{\partial \mathbf{x}} & \frac{\partial Q}{\partial t} \\ 0^T & 1 \end{pmatrix} \begin{pmatrix} \frac{\partial \mathbf{x}}{\partial \mathbf{X}} & \mathbf{v} \\ 0^T & 1 \end{pmatrix} \quad (3.18)$$

After multiplication one can obtain the well known equation between material and spatial time derivatives:

$$\frac{\partial Q^{**}}{\partial t} = \frac{\partial Q}{\partial t} + \frac{\partial Q}{\partial \mathbf{x}} \cdot \mathbf{v} \quad (3.19)$$

To ease notation we now recognize the material and spatial time derivatives $\frac{\partial Q^{**}}{\partial t} = \frac{\partial Q}{\partial t} \Big|_{\mathbf{x}}$, $\frac{\partial Q}{\partial t} = \frac{\partial Q}{\partial t} \Big|_{\mathbf{x}}$, and define the material and spatial derivatives the following way

$$\frac{d}{dt} := \frac{\partial}{\partial t} \Big|_{\mathbf{x}} \quad \frac{\partial}{\partial t} := \frac{\partial}{\partial t} \Big|_{\mathbf{x}}$$

The relation (3.19) can now be written in a form probably already known to the reader

$$\frac{dQ}{dt} = \frac{\partial Q}{\partial t} + (\mathbf{v} \cdot \nabla)Q \quad (3.20)$$

The next step is to relate the material and the referential description of the quantity, Q^{**} and Q^* respectively, by the mapping α . This relation is written as

$$Q^{**} = Q^* \circ \alpha^{-1}$$

By proceeding the exact same way as in (3.17) and (3.18) the relation between material and referential time derivatives is written

$$\frac{\partial Q^{**}}{\partial t} = \frac{\partial Q^*}{\partial t} + \frac{\partial Q^*}{\partial \chi} \cdot \hat{\mathbf{v}}$$

If we rather want to express the spatial derivative of Q^* in the spatial domain, we can use the definition of $\hat{\mathbf{v}}$ from equation (3.16) to end up with

$$\frac{\partial Q^{**}}{\partial t} = \frac{\partial Q^*}{\partial t} + \frac{\partial Q}{\partial \mathbf{x}} \cdot \mathbf{c}$$

This equation can be written in more common notation, and the following is known as *The fundamental ALE equation*

$$\frac{dQ}{dt} = \frac{\partial Q}{\partial t} \Big|_{\chi} + (\mathbf{c} \cdot \nabla)Q \quad (3.21)$$

and states that the time derivative in the material configuration equals its local (referential) derivative plus a convective term taking into account the relative difference in velocity between the two systems. It should be noted that the relations presented also holds for vector quantities.

Also, by combining equations (3.20) and (3.21), we can relate the spatial time derivative with the referential time derivative as

$$\frac{\partial Q}{\partial t} = \frac{\partial Q}{\partial t} \Big|_{\chi} - (\mathbf{w} \cdot \nabla)Q \quad (3.22)$$

3.6 Balance equations in the ALE framework

To obtain appropriate balance equations in the ALE framework we start by noting that the balance equations can be written in terms of the material derivatives as

$$\frac{d\rho}{dt} = \frac{\partial \rho}{\partial t} + \mathbf{v} \cdot \nabla \rho = -\rho \cdot \nabla \mathbf{v}$$

$$\rho \frac{d\mathbf{v}}{dt} = \rho \left(\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} \right) = \nabla \cdot \boldsymbol{\sigma}$$

By using equation (3.22), the spatial time derivatives can be replaced to obtain equations for the ALE framework. The following equations should hold in *any* time-dependent domain which does not necessarily need to coincide with the movement of material particles.

$$\begin{aligned} \frac{\partial \rho}{\partial t} \Big|_{\chi} + \mathbf{c} \cdot \nabla \rho &= -\rho \nabla \cdot \mathbf{v} \quad \text{in } \Omega^t \\ \rho \left(\frac{\partial \mathbf{v}}{\partial t} \Big|_{\chi} + (\mathbf{c} \cdot \nabla) \mathbf{v} \right) &= \nabla \cdot \boldsymbol{\sigma} \quad \text{in } \Omega^t \end{aligned}$$

Which shows that all one have to do in order to transform the Eulerian form of the balance equations into the ALE formulation is to replace the convective velocity with the relative velocity between the material and the mesh.

3.6.1 Mesh updating

The velocity \mathbf{w} can be seen as the mesh velocity when a computational mesh is used. In FSI, the ALE framework provides flexibility to combine the Lagrangian and Eulerian descriptions of motion. On the structural part, as well as on the fluid-structure interface, the domain Ω_s consists of the same material particles at all times and moves exactly with the material points within the structure. That is, $\mathbf{v}_s = \mathbf{w}_s$. The fluid domain has to follow the changes on the fluid-structure interface. Other than that the mesh velocity in the fluid domain is arbitrary in principle, but the choice of mesh velocity in the fluid domain is important for the accuracy of the solver. In general, important aspects to consider is distortion and squeeze of each element in the fluid domain.

A Laplacian smoothing algorithm is used to update the mesh, consisting of solving a Laplace (or Poisson) equation for the mesh displacement in the fluid. The method is a mesh regularization method where the lines in the mesh have equal potential. This method was first introduced by Winslow in 1963 [28].

3.7 Fluid Structure Interaction

By establishing the flexibility of the ALE formulation together with an equation for a moving domain we can now state the governing equations for the Fluid-Structure interaction problem. In the case of an Newtonian incompressible

fluid together with a linear elastic material in the absence of body forces, the mathematical problem consists of solving

$$\begin{aligned} \rho_f \left(\frac{\partial \mathbf{v}}{\partial t} + ((\mathbf{v} - \mathbf{w}) \cdot \nabla) \mathbf{v} \right) - \nabla \cdot \sigma_f(\mathbf{v}, p) &= 0 & \text{in } \Omega_f^t \\ \nabla \cdot \mathbf{v} &= 0 & \text{in } \Omega_f^t \\ \nabla^2 \mathbf{U} &= 0 & \text{in } \Omega_f^t \end{aligned}$$

$$\begin{aligned} \rho_s \frac{\partial \mathbf{v}}{\partial t} + \rho_s (\mathbf{v} \cdot \nabla) \mathbf{v} - \nabla \cdot \sigma_s(\mathbf{U}) &= 0 & \text{in } \Omega_s^t \\ \mathbf{w} &= \mathbf{v} & \text{in } \Omega_s^t \end{aligned}$$

In the case of fluid interacting with a solid structure, no material can cross the moving boundary Γ^t , and thus the fluid and solid velocity must be equal on the interface between the fluid and the solid. In general, one have to ensure mass conservation on the boundary. In addition the forces acting on the surface must be equal. A difference in forces from each material on the interface would result in infinite acceleration on the infinitely thin surface. Mathematically, these two conditions can be stated as

$$\left. \begin{aligned} \sigma_f \cdot \mathbf{n} &= \sigma_s \cdot \mathbf{n} \\ \mathbf{v}_f &= \mathbf{v}_s \end{aligned} \right\} \text{on } \Gamma^t$$

And will be used at all interfaces in FSI simulations. Kinematic or dynamic boundary conditions will be needed on the other boundaries as well, but will differ depending on the problem considered.

3.8 Coupling Fluid Flow with Poroelasticity

In a similar fashion to the Fluid-Structure interaction problem, and with the simplifications assuming the filtration velocity is small, the Biot problem together with Navier-Stokes equations consists of solving

$$\begin{aligned} \rho_f \left(\frac{\partial \mathbf{v}_f}{\partial t} + ((\mathbf{v}_f - \mathbf{w}) \cdot \nabla) \mathbf{v}_f \right) - \nabla \cdot \sigma_f(\mathbf{v}_f, p_f) &= 0 & \text{in } \Omega_f^t \\ \nabla \cdot \mathbf{v}_f &= 0 & \text{in } \Omega_f^t \\ \nabla^2 \mathbf{U} &= 0 & \text{in } \Omega_f^t \end{aligned}$$

$$\begin{aligned} \rho_p \frac{\partial \mathbf{v}_s}{\partial t} + \rho_p (\mathbf{v}_s \cdot \nabla) \mathbf{v}_s - \nabla \cdot \sigma_s(\mathbf{U}) + \nabla p_s &= 0 & \text{in } \Omega_s^t \\ \rho_p \frac{\partial \mathbf{v}_s}{\partial t} + \rho_p (\mathbf{v}_s \cdot \nabla) \mathbf{v}_s + K^{-1} \mathbf{q} + \nabla p_s &= 0 & \text{in } \Omega_s^t \\ \nabla \cdot (\mathbf{v}_s + \mathbf{q}) &= 0 & \text{in } \Omega_s^t \end{aligned}$$

Boundary conditions at the interface for the given problem have been discussed over the years, and in 1967 Beavers and Joseph [29] published a paper

describing experiments investigating the slip rate at a horizontal permeable wall located at $y=0$ with fluid flow above the porous medium. The boundary condition at the wall $y=0$ should be

$$\frac{\partial v_f}{\partial y} = \frac{\alpha_{BJ}}{K^{1/2}}(v_f - q), \quad (3.23)$$

where v_f is the fluid velocity tangential to the plane, q is the seepage or filtration velocity in the porous medium, K is the permeability and α_{BJ} is a constant depending on the material parameters of the porous medium close to the boundary. The derivative on the left hand side of equation (3.23) should be evaluated just above the plane and q should be evaluated just below the plane. In 1971 Saffmann [30] generalized the problem to other geometries as well as showing that the filtration velocity q could be left out of the equation only adding errors of order $O(K)$. Jones, (1973) [31] assumed shortly afterwards that the velocity jump was proportional to the shear stress rather than velocity shear. These additions to the original boundary condition proposed by Beavers and Joseph (known as the Beavers-Joseph-Saffmann (BSJ) condition) was used in spinal canal models by Drøsdal:

$$2\mathbf{n} \cdot \boldsymbol{\epsilon}(\mathbf{v}_f) \cdot \boldsymbol{\tau} = \alpha_{BJ} K^{-1/2} \mathbf{v}_f \cdot \boldsymbol{\tau}$$

and showed to have minimal effect on key value measurements such as filtration velocity in the cord as well as fluid velocity in both the SAS and inside the syrinx. (Values changed by $10^{-4}\%$ for α_{BJ} ranging from 0 to 1). This condition is therefore omitted in this thesis.

As in the FSI case, we assume conservation of mass and continuity of stresses at the interface. The boundary condition is related through the velocities and the Darcy flux at the boundary.

$$\left. \begin{aligned} \mathbf{v}_f &= \mathbf{v}_s + \mathbf{q} \\ \sigma_f \cdot \mathbf{n} &= \sigma_s \cdot \mathbf{n} - p_p \mathbf{n} \end{aligned} \right\} \text{on } \Gamma^t \quad (3.24)$$

Strictly speaking, mass conservation does not necessarily imply the tangential velocities to be continuous at the interface, and the transition in tangential velocity is what the BJS-condition tries to capture. However, as we shall see later, with the limitations by using continuous functions in the computational modeling the tangential velocities will also have to be continuous. This limitation is also the case for the pressure, and as pointed out by Nield and Bejan (2013) [22] these assumptions together with the BJS-condition results in an overdetermined system. The pressure will be continuous on the microscopic scale, but only approximately continuous on the macroscopic (averaging) scale. Despite these limitations, we stick with the boundary conditions (3.24) assuming, as showed on porous flow, the BJS-condition barely alters the solution.

Chapter 4

Numerical Methods

4.1 The Finite Element Method

The theory presented in this section is partly inspired by the works of Langtangen in "Finite Element Method - INF5620 lecture notes" [32]

Consider the Poisson-equation

$$-\nabla^2 v = f \quad \text{in } \Omega \quad (4.1)$$

$$v = v_0 \quad \text{on } \partial\Omega_D \quad (4.2)$$

$$\frac{\partial v}{\partial n} = g \quad \text{on } \partial\Omega_N \quad (4.3)$$

where $\Omega \in \mathbb{R}^d$ is a domain, $v = v(x)$ is an unknown function and f is a source function. The boundary, $\partial\Omega$ is divided into two parts. $\partial\Omega_D$ for the Dirichlet boundary condition, and $\partial\Omega_N$ for the Neumann condition.

4.1.1 Variational formulation

(4.1) is known as the strong form of the equation. To reformulate the problem and state a weak formulation we multiply the equation with a test function, $\phi \in \hat{V}$, where \hat{V} is some function space, and integrate over the domain. Weak formulations are important in the sense that differential equations can be transformed into systems of linear equations. In the rest of this text the following notation is used for the inner product of two functions

$$(v, \phi)_\Omega = \int_\Omega v \phi \, dx$$

By multiplying (4.1) with a test function, ϕ and integrating over the domain, the weak form is obtained

$$(\nabla^2 v + f, \phi)_\Omega = 0 \quad \forall \phi \in \hat{V}$$

We are now searching for a v to satisfy the weak form instead of the strong. This equation should hold for all ϕ in the function space \hat{V} . The trial function does not necessarily have to lie in the same function space, in general $v \in V$.

In this thesis we will use two Sobolov spaces (named after the Russian mathematician Sergei Sobolov) widely used in Finite Element computing. For these definitions to be valid, we assume that the functions v are all **—locally integrable** and in the case of definition (4.2), has one **weak derivative**. For more details on weak derivatives and the generalized concept of Sobolev spaces and functional analysis, we refer to the textbook by Brenner and Scott [33].

Definition 4.1 *Let Ω be an open subset of \mathbb{R} with a piecewise smooth boundary. We then define the L^2 -norm as follows*

$$\|v\|_{L^2(\Omega)} = (\int_{\Omega} v^2 dx)^{\frac{1}{2}}$$

The corresponding L^2 -space is defined via

$$L^2(\Omega) = \{v : \Omega \rightarrow \mathbb{R} \mid \int_{\Omega} v^2 dx < \infty\}$$

Definition 4.2 *Let Ω be an open subset of \mathbb{R} with a piecewise smooth boundary. We then define the H^1 -norm as follows*

$$\|v\|_{H^1(\Omega)} = (\int_{\Omega} [v^2 + (\nabla v)^2] dx)^{\frac{1}{2}}$$

The corresponding H^1 -space is defined via

$$H^1(\Omega) = \{v : \Omega \rightarrow \mathbb{R} \mid \int_{\Omega} [v^2 + (\nabla v)^2] dx < \infty\}$$

In other words, using functions from these spaces, we can have some assurance that the integrals involved in the variational form are bounded. By the divergence theorem, a generalized concept of integration by parts we can state the variational problem as follows: find $v \in V$ such that

$$(\nabla v, \nabla \phi)_{\Omega} = (f, \phi)_{\Omega} + (g, \phi)_{\partial\Omega_N} \quad \forall \phi \in \hat{V} \quad (4.4)$$

Where we have used that $\frac{\partial v}{\partial n} = g$ on $\partial\Omega_N$. (4.4) is known as the variational formulation of the Poisson problem. The left hand side is known as the bilinear form while the right hand side is the linear form. In generic form the equation can be written

$$a(v, \phi) = L(\phi) \quad (4.5)$$

The first derivative of v appears in the variational form. A common choice is then

$$\begin{aligned} V &:= \{v \in H^1(\Omega) : v = v_0 \text{ on } \partial\Omega_D\} \\ \hat{V} &:= \{v \in H^1(\Omega) : v = 0 \text{ on } \partial\Omega_D\} \end{aligned}$$

In other words, the trial and test functions are in the same function space, except on the boundary.

4.1.2 Finite elements

The next step is to approximate v with a sum of basisfunctions in the finite-dimensional function space, $V = \text{span}\{\phi_0, \phi_1, \dots, \phi_N\}$. Here, ϕ_i represents the basis functions and we search for a solution $v_h \in V$ such that v_h can be written as a linear combination of the basis functions. The first step in the finite element method consists of dividing the domain into smaller parts

$$\Omega = \Omega_0 \cup \Omega_1 \cup \dots \cup \Omega_{N_e}$$

where N_e is the number of elements. Each element have a number of nodes within them depending on what type of basis functions to be used. Let's first consider the continuous Galerkin basis functions, in a one-dimensional domain. There is exactly one basisfunction for each node located at x_i . These basis functions have the property that

$$\phi_i(x_j) = \begin{cases} 1 & \text{for } i = j \\ 0 & \text{for } i \neq j \end{cases}$$

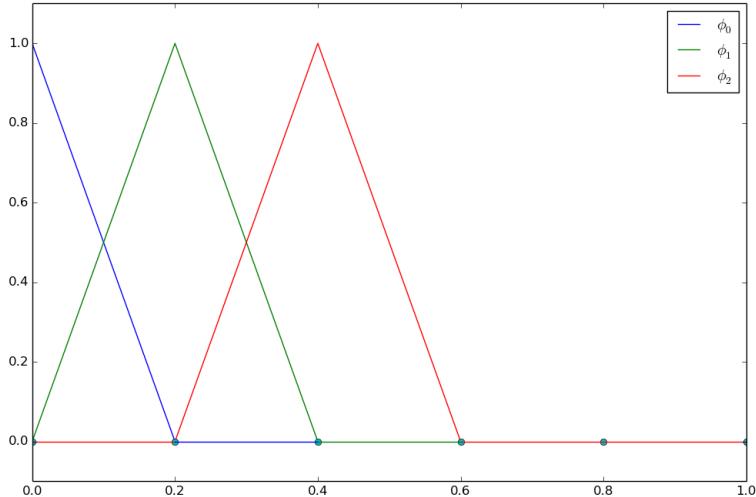


Figure 4.1: The three first linear basis functions on the unit interval divided uniformly into 5 elements, $\Omega_0 = [0, 0.2]$, $\Omega_1 = [0.2, 0.4]$ and so on. In the case of Dirichlet Boundary conditions at $x = 0$, ϕ_0 will not be included in the function space

That is, the basis functions ϕ_i are zero on all nodes except at node i . Each basis function is constructed by taking the Lagrange-polynomial which is 1 at the given node and 0 on the neighboring nodes. Note that the basis functions are two Lagrange-polynomials "pieced together" at the node where it's value is 1. For the rest of the domain the basis functions are defined to be 0.

Now, let's return to the original problem (4.1)-(4.3) in scalar form. We start by approximating v as a linear combination of all the basis functions.

$$v_h = \sum_{i=0}^N c_i \phi_i$$

The definitions of v_h and V now give rise to a linear system. Using the Einstein summation convention, $x_i y_i = \sum_0^N x_i y_i$, the discretized version of (4.4) is now written

$$-c_i (\nabla \phi_i, \nabla \phi_j)_\Omega = (f, \phi_j)_\Omega - (g, \phi_j)_{\partial \Omega_N}$$

In the case of Dirichlet boundary conditions all test functions ϕ_j will take the value 0 on $\partial \Omega_D$ and the linear system will be adjusted to take these boundary conditions into account.

The system can be written in matrix form, and in the end the problem consists of solving the linear system

$$A_{i,j} c_i = b_j$$

4.2 The FEniCS software

When the variational form has been carried out, implementation in FEniCS is relatively simple. The programs in this study are written in the Python programming language. When programming with Python, we first need to import dolfin to access the DOLFIN library, containing classes convenient and efficient for finite element computing. In Python the full library can be imported as simple as

```
from dolfin import *
```

Now, let's focus our attention on solving the following problem:

$$\begin{aligned} \nabla^2 v &= 20x & \text{in } \Omega \\ v(0, y) &= 0, \quad v(1, y) = 1 \\ \frac{\partial v(x, 0)}{\partial n} &= \frac{\partial v(x, 1)}{\partial n} = 0 \end{aligned}$$

Where Ω is the unit square, $\Omega = [0, 1] \times [0, 1]$

First of all we need to define the computational mesh.

```
mesh = UnitSquareMesh(10,10)
```

The class `UnitSquareMesh` initializes a mesh with triangular cells. The mesh consists of $n \times m$ squares depending on the arguments, n and m , sent into the constructor. Each of these squares are divided on the diagonal to form two triangles, and these triangles are the computational cells. In this case we get the unit square divided into 10×10 smaller squares and thus the total number of triangles, or cells, will be 200.

The next thing to do is to define an appropriate function space for the test functions. The solution will be a linear combination of these functions and will be in (almost) the same function space.

```
V = FunctionSpace(mesh, 'CG', 1)
```

The function space needs a domain, type of element, and the degree of the element. In this case we use Continuous Galerkin elements ('CG') with degree 1. These basis functions are visualized in Figure 4.1

We can then define our test- and trial functions v and ϕ

```
v = TrialFunction(V)
phi = TestFunction(V)
```

Note that the test- and trial function seem to be in the exact same function space. This is the case except when imposing Dirichlet boundary conditions. The functions f , v_0 and g can be defined by using the 'Constant' or 'Expression' classes. We set $f = 20x$ and use Dirichlet boundary conditions, $v(0, y) = 0$, $v(1, y) = 1$ and Neumann conditions $\frac{\partial v(x,0)}{\partial n} = \frac{\partial v(x,1)}{\partial n} = 0$. The homogenous Neumann condition is simple in the finite element method as the terms appearing after integration by parts can be dropped. If this is not the case, we can insert g for $\frac{\partial u}{\partial n}$ on the boundary integral appearing in the variational form. When the Neumann conditions are incorporated this way we say that the boundary conditions are weakly imposed. Functions (or classes) describing the boundaries must also be defined:

```
f = Expression('20*x[0]')
def boundary0(x, on_bnd):
    return on_bnd and near(x[0], 0.0)
def boundary1(x, on_bnd):
    return on_bnd and near(x[0], 1.0)

bc0 = DirichletBC(V, 0.0, boundary0)
bc1 = DirichletBC(V, 1.0, boundary1)
bcs = [bc0, bc1]
```

Note that ' $x[0]$ ' means first dimension in space, ' $x[1]$ ' means second dimension and so on. The Dirichlet conditions are put in a list. Next, the variational form is defined, and when solving for a function v , the boundary conditions are added to the "magic" `solve` function.

```

F = inner(grad(v),grad(phi))*dx - inner(f,phi)*dx
v = Function(V)
solve(lhs(F)==rhs(F),v,bcs)
plot(v)

```

The functions `lhs` and `rhs` separates the form `F` into the left hand side, equivalent to the bilinear form and to the right hand side, equivalent to the linear form. Specifying the full form `F` is simple and convenient when the equations are short and simple. If we want to relate the code to the mathematics as written in (4.5) we can, define these forms manually.

```

a = inner(grad(v),grad(phi))*dx
L = inner(f,phi)*dx
v = Function(V)
solve(a==L,v,bcs)

```

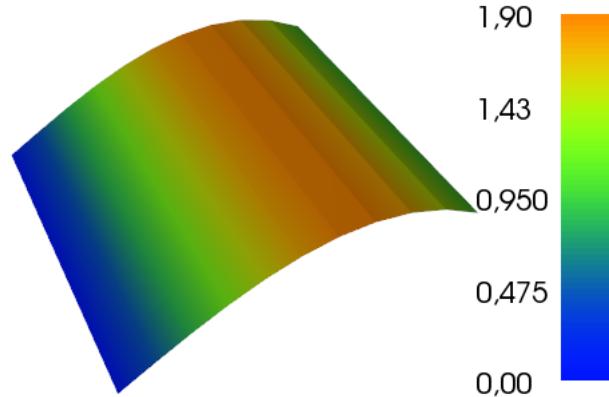


Figure 4.2: Plot of the computed solution

The plot in the figure is slightly rotated. The solution is independent of y -position, as expected.

4.3 A Discontinuous Galerkin method

Discontinuous Galerkin (DG) methods is a relatively new tool for CFD simulations. The method itself was developed during the 1970s and has been used increasingly the last few decades. Unlike with the Continuous Galerkin elements, we now allow the solution to be discontinuous, i.e. cells do not share nodes anymore. Instead of solving over the whole domain, we now seek approximate continuous solutions on each cell independently of the others. To this end it is convenient to define the average and jump of a discontinuous variable

$$\{\mathbf{v}\} = \frac{1}{2}(\mathbf{v}^+ + \mathbf{v}^-) \quad [\mathbf{v}] = \mathbf{v}^+ - \mathbf{v}^-$$

where \mathbf{v}^+ and \mathbf{v}^- is the solution at two neighboring cells at cell E^+ and E^- . These definitions can be accessed easily in FEniCS as

```
avg(v)
jump(v)
```

The normal vectors are denoted \mathbf{n}^+ and \mathbf{n}^- . If consistent, the choice of \mathbf{n} is arbitrary. [34]

It should be noted that *the weak form presented here are not used in the spinal cord simulation in this thesis* as there were problems with the correct boundary conditions at the interface. The reason for including a short discussion about a DG-method lies in the advantage of having functions discontinuous over the interface between the fluid and the solid, especially in the Biot problem. For instance, if a function \mathbf{v} should describe the fluid velocity in the fluid, and the structural velocity in the solid for the Biot problem, this variable would have to be discontinuous over the interface. FEniCS does not currently have the possibility to have discontinuous elements over the interface only, and therefore the function would have to be considered discontinuous over the whole domain.

4.3.1 Stokes flow

Consider the following problem in $\Omega = [0, 1] \times [0, 1]$:

$$\begin{aligned} \nabla \cdot \sigma &= 0 \text{ in } \Omega \\ \nabla \cdot \mathbf{v} &= 0 \text{ in } \Omega \\ \mathbf{v} &= 0 \text{ on } \Omega_w \\ p\mathbf{n} - \mu \frac{\partial \mathbf{v}}{\partial n} &= p_{in}\mathbf{n} \text{ on } \Omega_{in} \\ p\mathbf{n} - \mu \frac{\partial \mathbf{v}}{\partial n} &= p_{out}\mathbf{n} \text{ on } \Omega_{out} \end{aligned}$$

where σ is the incompressible Newtonian fluid stress tensor as previously described. The inlet is at $x=0$, while the outlet is at $x=1$. The rigid walls are situated where $y=0$ and $y=1$. By choosing $p_{in} = 2$, $p_{out} = 0$, a steady Poiseuille flow should be obtained. The boundary condition on the inlet and outlet are Pseudo-traction as described in chapter 3. The equations describes what is often known as Stokes flow.

By integrating Stokes equation by parts, adding symmetry and penalty terms as explained by Rivière and Yotov [35] we end up with a consistent DG-method. Since the method involves many terms we derive the weak formulation term by term. The set of all elements are denoted as e , interior facets as Γ and exterior facets as $\partial\Omega$. Starting with the diffusion term in divergence form, we multiply

with a test function and integrate by parts:

$$\begin{aligned}
& 2\mu \sum_{E \in e} (\epsilon(\mathbf{v}), \epsilon(\Phi))_E - 2\mu \sum_{E \in \Gamma} (\{\epsilon(\mathbf{v})\} \cdot \mathbf{n}_e, [\nabla \Phi])_E - 2\mu \sum_{E \in \Gamma} (\{\epsilon(\Phi)\} \cdot \mathbf{n}_e, [\nabla \mathbf{v}])_E \\
& + \frac{\beta}{h} \sum_{E \in \Gamma} ([\mathbf{v}], [\Phi])_E - 2\mu \sum_{E \in \partial\Omega} (\Phi, \epsilon(\mathbf{v}) \cdot \mathbf{n}_e)_E \\
& - 2\mu \sum_{E \in \partial\Omega} (\mathbf{v}, \epsilon(\Phi) \cdot \mathbf{n}_e)_E + \frac{\beta}{h} \sum_{E \in \partial\Omega} (\mathbf{v}, \Phi)_E
\end{aligned}$$

Similiarly, the contributions from the term $-\nabla p$ will be

$$-\sum_{E \in e} (p, \nabla \cdot \Phi)_E + \sum_{E \in \Gamma} (\{p\}, [\Phi] \cdot \mathbf{n}) + \sum_{E \in \Gamma} (\{\eta\}, [\mathbf{v}] \cdot \mathbf{n})$$

With $DG_1 \times DG_0$ elements for velocity and pressure the following results were obtained with the convergence rate computed as

$$\frac{\ln(\frac{e^{n+1}}{e^n})}{\ln(\frac{h^{n+1}}{h^n})},$$

N representing the number of cells in the $N \times N$ unit square and h computed in FEniCS as `mesh.hmin()` for each refinement

N	dofs	$\ v - v_h\ _{L^2}$	$\ p - p_h\ _{L^2}$	rate u	rate p
4	224	1.8318e-02	1.5590e-01	—	—
8	896	1.0732e-02	7.6566e-02	0.7713	1.0259
16	3584	5.8110e-03	3.7890e-02	0.8851	1.0149
32	14336	3.0221e-03	1.8895e-02	0.9432	1.0038
64	57344	1.5410e-03	9.4481e-03	0.9717	0.9999
128	229376	7.7808e-04	4.7267e-03	0.9859	0.9992

4.4 Womersley Flow

In the cardiovascular system, pressure pulses travel along different blood vessels such as veins, capillaries and the aorta. These types of pulsating flows in tubes or channels have been given the name Womersley flow. The characteristics and velocity profile of the flow depends of several parameters such as the length of the tube or channel, pulsation frequency and fluid properties. In the end, the ratio between transient inertial forces and viscous forces is the fundamental difference separating flow patterns in pulsating flow. To this end the Womersley number, α is defined as follows:

$$\alpha^2 = \frac{\text{transient inertial force}}{\text{viscous force}} = \frac{\rho \omega V}{\mu V L^{-2}} = \frac{L^2 \omega \rho}{\mu}$$

Where, ρ is the fluid denisty, μ is the dynamic viscosity, L is a length scale and ω is the pulsation frequency. In the 2D-model presented here, the modeling

of SAS around the spinal cord consists of two channels where the Womersley number

$$\alpha = L \left(\frac{\omega \rho}{\mu} \right)^{\frac{1}{2}},$$

would have a large impact on the flow characteristics. A low Womersley number (typically $\alpha < 1$ means the frequency is relatively low so the flow develops a velocity profile close to a parabola at each cycle. When α is large ($\alpha > 10$) the inertial forces dominate and more complicated phenomena such as bidirectional flow, i.e. flow in opposite directions over a cross section, could occur.

Even though the flow should be aligned with the channel due to the incompressibility constraint, pulsating flow tends to give rise to some horizontal or radial flow with the boundary conditions previously described due to numerical errors. We will return to this issue in the next subsection. For now we assume the following conditions to hold

$$\begin{aligned} \mathbf{v} \cdot \boldsymbol{\tau} &= 0 \text{ on } \partial\Omega_{\text{in}} \\ \mathbf{v} \cdot \boldsymbol{\tau} &= 0 \text{ on } \partial\Omega_{\text{out}} \end{aligned} \quad (4.6)$$

Where $\boldsymbol{\tau}$ is the tangential vector on the boundary surface.

The velocity and pressure should therefore satisfy

$$\begin{aligned} \frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} &= \nabla \cdot \boldsymbol{\sigma} && \text{in } \Omega, \\ \nabla \cdot \mathbf{u} &= 0 && \text{in } \Omega, \\ \mathbf{v} &= 0 && \text{on } \Omega_w, \\ \mu \frac{\partial \mathbf{v}}{\partial n} - p \mathbf{n} &= p_{\text{in}}(t) \mathbf{n} \text{ and } \mathbf{v} \cdot \boldsymbol{\tau} = 0 && \text{on } \Omega_{\text{in}}, \\ \mu \frac{\partial \mathbf{v}}{\partial n} - p \mathbf{n} &= p_{\text{out}}(t) \mathbf{n} \text{ and } \mathbf{v} \cdot \boldsymbol{\tau} = 0 && \text{on } \Omega_{\text{out}}. \end{aligned}$$

Exact solutions exists to both the channel and pipe cases. Langlois and Deville [36] have derived several solutions to equations of viscous flow, including channel flow with a pulsatile pressure gradient as above. We now add a oscillating pressure gradient so that

$$-\frac{1}{\rho} \frac{\partial p}{\partial x} = -C \cos(\omega t)$$

where C is a constant describing the strength of the pulse. Due to the assumption of axial flow only, the momentum equation in x-direction gives:

$$\frac{\partial v_1}{\partial t} = -\frac{1}{\rho} \frac{\partial p}{\partial x_1} + \nu \frac{\partial^2 v_1}{\partial x_2^2} \quad (4.7)$$

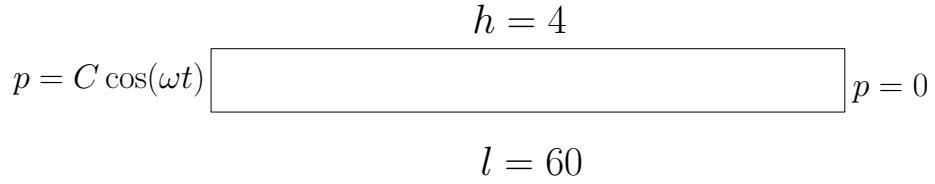


Figure 4.3: Schematic of Womersley flow with dimensions as in the spinal cord

The presented solution did not correspond to the simulations at all, therefore an opportunity to utilize the symbolic python package sympy arised. The solution presented by Langlois and Deville was

$$v_1 = -\frac{C}{\omega} \left[\left(1 - \frac{f_1(\omega, x_3)}{f_3(kh)} \right) \sin(\omega t) - \frac{f_2(\omega, x_3)}{f_3(kh)} \cos(\omega t) \right]$$

where

$$\begin{aligned} k &= \sqrt{\frac{\omega}{2\nu}} \\ cc(x) &= \cos(x) \cosh(x) \\ ss(x) &= \sin(x) \sinh(x) \\ f_1(\omega, x_3) &= cc(kx_3)cc(kh) + ss(kx_3)ss(kh) \\ f_2(\omega, x_3) &= cc(kx_3)ss(kh) - ss(kx_3)cc(kh) \\ f_3(\omega) &= cc^2(\omega) + ss^2(\omega) \end{aligned}$$

In sympy, even a complicated solution like this is easy to verify. The package lets the user define symbols to work with. Regular multiplication and general python functions can also be used.

```
from sympy import *
x3, C, x, h, t, w, nu = symbols('x3 C x h t w nu')
k = sqrt(w/(2*nu))
def cc(x):
    return cos(x)*cosh(x)
.
.
.

f3 = cc(w)**2 + ss(w)**2

u = -C/w*((1-f1/f3)*sin(w*t) - f2/f3*cos(w*t))    # presented solution
d2u = nu*diff(diff(u,x3),x3)
dt = diff(u,t)

print simplify(d2u-dt)
```

From (4.7) the code snippet should print the pressure gradient divided by the density

$$\frac{1}{\rho} \frac{\partial p}{\partial x_3} = C \cos(\omega t)$$

This was not the case. Some error with the sign of the solution must have been made.

The solution

$$v_1 = -\frac{C}{\omega} \left[\left(1 - \frac{f_1(\omega, x_3)}{f_3(kh)} \right) \sin(\omega t) + \frac{f_2(\omega, x_3)}{f_3(kh)} \cos(\omega t) \right]$$

Where the sign before the cos-term has been changed, yielded the correct pressure gradient, and hence this solution was used when error-estimates were investigated.

4.4.1 A penalty method for the boundary conditions

To understand the problems arising with Neumann Conditions on both inlet and outlet we first consider the convection-diffusion equation.

$$\begin{aligned} -\mu \nabla^2 u + \mathbf{v} \cdot \nabla u &= f \quad \text{in } \Omega \\ u &= u_0 \quad \text{on } \partial\Omega_D \\ \mu \frac{\partial u}{\partial n} &= g \quad \text{on } \partial\Omega_N \end{aligned}$$

Where u is an unknown function, \mathbf{v} is a prescribed velocity, μ is a diffusion constant and f is a source term. The weak formulation reads:

Find $u \in H_{u_0}^1$

$$\mu(\nabla u, \nabla \phi)_\Omega + (\mathbf{v} \cdot \nabla u, \phi)_\Omega = (f, \phi)_\Omega + (g, \phi)_{\partial\Omega_N} \quad \text{for all } \phi \in H_0^1.$$

Where the subscript on the two H-spaces indicate the values for functions on $\partial\Omega_D$ for these spaces.

What has not been mentioned so far is the existence and uniqueness of the finite element solutions. To establish some concepts addressing these questions we let V denote a function space (in particular, V is a Hilbert space which has to satisfy given conditions on the associated inner product), where the inner product is denoted $(\cdot, \cdot)_V$ and the norm $\|\cdot\|_V$, (see e.g. Elman et al [37]) and define the following

Definition 4.3 (Coercivity)

A bilinear form $a(\cdot, \cdot)$ is said to be coercive with respect to the norm $\|\cdot\|_V$ if there is a positive constant γ such that

$$a(u, u) \geq \gamma \|u\|_V^2 \quad \text{for all } u \in V$$

Definition 4.4 (Continuity)

A bilinear form $a(\cdot, \cdot)$ is continuous with respect to the norm $\|\cdot\|_V$ if there is a positive constant Γ such that

$$a(u, u) \leq \Gamma \|u\|_V \|\phi\|_V \quad \text{for all } u, \phi \in V$$

A linear functional $L(\phi)$ is continuous with respect to $\|\cdot\|_V$ if there is a constant Λ such that

$$L(\phi) \leq \Gamma \|\phi\|_V \text{ for all } \phi \in V$$

In order to have a well posed problem, and ensure the existence of a unique solution $u \in V$ satisfying

$$a(u, \phi) = L(\phi) \text{ for all } \phi \in V$$

$a(\cdot, \cdot)$ and $L(\cdot)$ have to satisfy these criteria. This is commonly known as the Lax-Milgram lemma.

The convection term

$$c(u, \phi) = (\mathbf{v} \cdot \nabla u, \phi)_\Omega \quad (4.8)$$

makes this problem very different from the Poisson-problem (4.1), (4.2), (4.3), used when introducing the finite element method. In the light of the previous definitions we apply Green's theorem to (4.8)

$$\begin{aligned} c(u, \phi) &= \int_{\Omega} \phi \mathbf{v} \cdot \nabla u \, dx \\ &= - \int_{\Omega} u \mathbf{v} \cdot \nabla \phi \, dx - \int_{\Omega} u \phi \nabla \cdot \mathbf{v} \, dx + \int_{\partial\Omega_N} u \phi \mathbf{v} \cdot \mathbf{n} \, dS \\ &= -c(\phi, u) + \int_{\partial\Omega_N} u \phi \mathbf{v} \cdot \mathbf{n} \, dS \end{aligned}$$

Where the last step follows from the assumption of incompressibility. From this

$$c(u, u) = \frac{1}{2} \int_{\Omega_N} u^2 \mathbf{v} \cdot \mathbf{n} \, dS$$

It is obvious that a Neumann condition on the inflow boundary, where \mathbf{v} and \mathbf{n} points in opposite directions, makes a negative contribution to the bilinear form $a(u, u)$. Therefore coercivity can not be completely ensured, and it typically depends on the magnitude of the velocity at the inlet.

In several problems, prescribing Neumann conditions are simpler than providing inlet velocities. The pressure is set as a scalar value and can be assumed to have no spatial variation over the boundaries if the model has flat surfaces with normals aligned with the longitudinal axis on the inlet and outlet.

Barth and Carey [38] have described a penalty method for imposing the boundary conditions (4.6). This was needed as horizontal flow at, and close to the boundaries caused problems interacting with the elastic spinal cord. The penalty method consists of adding a term at the relevant boundaries, penalizing parts where the boundary condition do not hold. For the velocity, we want $\mathbf{v} - (\mathbf{v} \cdot \mathbf{n})\mathbf{n}$ on the boundary, therefore the first variation of the least squares penalty functional is added to the variational formulation. This has been done in all simulations with Neumann boundary conditions at the inlet in this thesis. The penalty functional is given as

$$I(\mathbf{v}) = \frac{1}{2\epsilon} \int_{\partial\Omega_p} [\mathbf{v} - (\mathbf{v} \cdot \mathbf{n})\mathbf{n}] \cdot [\mathbf{v} - (\mathbf{v} \cdot \mathbf{n})\mathbf{n}] \, dS,$$

where $0 < \epsilon \ll 1$ is the penalty parameter. The contribution from the first variation will be

$$I'(\mathbf{v})(\Phi) = \frac{1}{\epsilon} \int_{\partial\Omega_p} [\mathbf{v} - (\mathbf{v} \cdot \mathbf{n})\mathbf{n}] \cdot [\Phi - (\Phi \cdot \mathbf{n})\mathbf{n}] dS.$$

This term will clearly contribute positively to the bilinear form and the penalty parameter should ensure that this contribution is large. The method is tested on the coupled CFD solver, first with one time step of the Backwards Scheme and gradually refinement of the rectangular mesh. The time step is small, so the error introduced by the time discretization is negligible.

N	dofs	$\ v - v_h\ _{L^2}$	rate	$\ v - v_h\ _{H^1}$	rate
4	187	1.00e+00	—	6.65e+00	—
8	659	1.50e-01	2.739	1.95e+00	1.768
16	2467	1.91e-02	2.977	4.95e-01	1.981
32	9539	2.39e-03	2.997	1.24e-01	1.997
64	37507	2.99e-04	2.998	3.10e-02	1.999

Table 4.1: Errors P2-P1 elements

N	dofs	$\ v - v_h\ _{L^2}$	rate	$\ v - v_h\ _{H^1}$	rate
4	419	1.42e+01	—	1.43e+02	—
8	1539	7.84e-01	4.180	1.57e+01	3.186
16	5891	4.75e-02	4.045	1.90e+00	3.047
32	23043	2.97e-03	3.999	2.38e-01	3.000
64	91139	1.86e-04	3.999	2.97e-02	2.999

Table 4.2: Errors P3-P2 elements

C = 1000
dt = 0.001
T = 0.1
Backward Euler 1st order

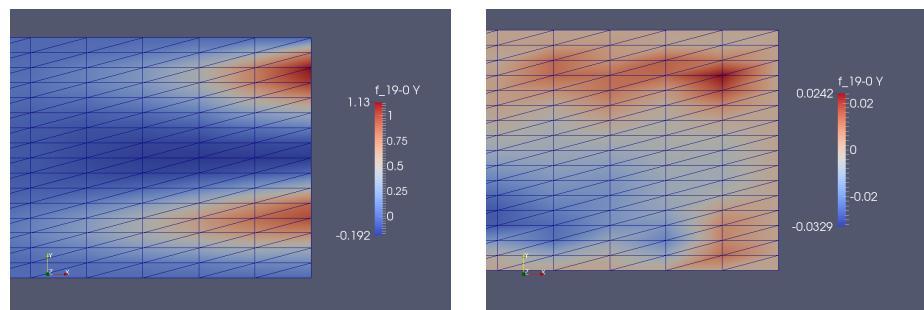


Figure 4.4: Velocity distribution in y -direction at $t=0.1$ without (left) and with the penalty method at the outlet. These velocities are of several orders lower than in the x -direction, but accumulates over time without the penalty method

Chapter 5

Numerical methods for FSI

5.1 A benchmark FSI-problem

Within CFD, a benchmark is a configuration or a test case which should help test and compare different numerical methods and code implementations. A classical Fluid Dynamics problem regarding flow around a circular cylinder has been under vast research the last 50 years, working as a test case for both laminar and turbulent flows. One of the most cited benchmark proposals for this case is the problem described by Michael Schafer et. al in 1996 [39]. The research group still focus on these kinds of problems and one of the co-authors of the 1996 paper, Stefan Turek, together with Jaroslav Hron has proposed a similar benchmark for FSI solvers, consisting of the exact same domain and rigid cylinder, but now with an elastic flag attached to it [40].

The first results presented will contain a validation of the present FSI-solver implemented in FEniCS compared to the results of Turek and Hron in their benchmark proposal.

A proper validation of a FSI solver requires separate verification of the fluid and structural parts as well as coupled tests. In the present study we solve the system of equations with a *monolithic* approach, i.e. full coupling between the fluid and solid. The alternative would be a *partitioned* approach, where the fluid and solid equations are solved separately. For instance, one can solve the fluid equations independently and then proceed by solving the solid equation with prescribed stress on the interface computed from the fluid solution. Iteration back and forth would be needed until convergence.

The fully coupled monolithic scheme is usually preferred with respect to accuracy and stability. Also, when the systems are strongly coupled in nature, i.e. the solid movement is affected by the fluid movement and vice versa, a monolithic scheme would be advantageous. The partitioned approach, on the other hand, can benefit from numerous previous studies where efficiency and stability for various solution techniques have been investigated. See e.g. [41] for a short review. In addition, solving many smaller matrix systems will be way faster than solving one large system with the same number of unknowns.

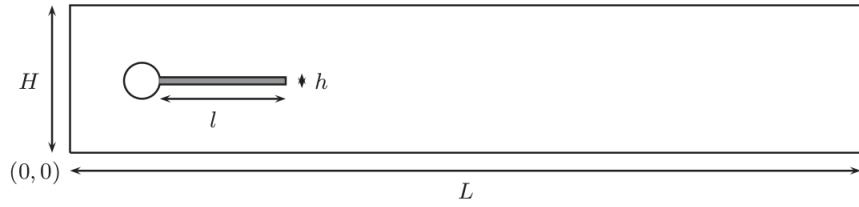


Fig. 1. Computational domain

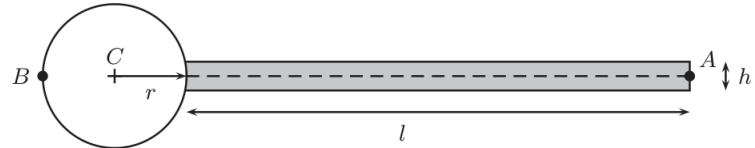


Fig. 2. Detail of the structure part

Figure 5.1: The Domain as published in [40]

5.1.1 Domain, Initial- and boundary conditions

The origin is set at the bottom left corner. We also set:

- The channel height, $H = 0.41$
- The channel length, $L = 2.5$
- The circle center $C = (0.2, 0.2)$
- The right bottom corner of the elastic structure has position $(0.6, 0.19)$
- The elastic structure has length, $l=0.35$ and height $h=0.02$
- At the left boundary, the inlet, of the channel, we set a prescribed parabolic velocity profile

$$\mathbf{v}_{\text{in}}(0, y) = 1.5\bar{v}_0 \frac{y(H-y)}{(\frac{H}{2})^2}$$

- In the case of unsteady flow a smooth increase in time is used:

$$\mathbf{v}_{\text{in}}(t, 0, y) = \begin{cases} \mathbf{v}_{\text{in}}(0, y) \frac{1-\cos(\frac{\pi}{2}t)}{2} & \text{if } t < 2.0 \\ \mathbf{v}_{\text{in}}(0, y) & \text{otherwise} \end{cases}$$

- On the outlet, the condition $\sigma \cdot \mathbf{n} = 0$ is applied - On rigid walls the no-slip condition is used - On the interface, Γ^t , the previously described coupling conditions are applied:

$$\left. \begin{array}{l} \sigma_f \cdot \mathbf{n} = \sigma_s \cdot \mathbf{n} \\ \mathbf{v}_f = \mathbf{v}_s \end{array} \right\} \text{on } \Gamma^t$$

In addition to the interface, Γ^t , it may be convenient to define the fluid boundary, $\partial\Omega_f^t$ consisting of the outer rectangle and the part of the circle in contact with the fluid, and the solid boundary, $\partial\Omega_s^t$ consisting of the circle in contact with the solid.

5.2 Fluid Structure Interaction using the Finite Element method

A solver was implemented from scratch in Python using the DOLFIN library. FSI solvers within the FEniCS framework exists, e.g. under Unicorn or CBC.twist. For instance, Selin [42] implemented a partitioned solver in FEniCS in his PhD-thesis, using the already existing modules for solving fluid flow and structural deformations separately (CBC.Flow and CBC.Twist). With the implementation presented here, it will be easier to adjust the solver with respect to the model. The equations can be changed whether we want the spinal cord to be porous, elastic, poroelastic or viscoelastic. The use of two separate solvers is also problematic for the monolithic approach, when all equations should be solved simultaneously. On the other hand, solvers implemented by experienced and skilled research groups will probably be a lot more efficient and should already have been validated.

In the rest of this section, we give a brief explanation to the mathematics and implementation in FEniCS. In the previous example we saw the close link between code and mathematics. :

$$\begin{aligned} a(\mathbf{v}, \Phi) &= (\nabla \mathbf{v}, \nabla \Phi)_{\Omega} \\ L(\mathbf{v}) &= (\mathbf{f}, \Phi)_{\Omega} \end{aligned}$$

Translates to

```
a = inner(grad(v), grad(phi))*dx
L = inner(f, phi)*dx
```

In deriving variational forms, we try to keep this close link by recalling the symbols used for velocity (\mathbf{v}), total displacement (\mathbf{U}), pressure (p), and mesh velocity, (\mathbf{w}). Also recall that in the solid the mesh moves exactly with the velocity of the structure so $\mathbf{w}_s = \mathbf{v}_s$

5.2.1 Temporal discretization

In order not to overload this thesis with notation and superscripts, we have used the notation $\mathbf{v} := \mathbf{v}^{n+1}$ to denote the value of a function at the next time-step. Similarly, we define $\mathbf{v}^{(1)} := \mathbf{v}^n$ to denote the (known) value of a function at the present time step.

The total displacement \mathbf{U} can be expressed as a function of the displacement from the previous time step, $\mathbf{U}^{(1)}$, and the mesh velocity \mathbf{w} . We have used an implicit scheme in time, i.e, $(\frac{\partial \mathbf{v}}{\partial t})^{n+1} \approx \frac{\mathbf{v}^{n+1} - \mathbf{v}^n}{\Delta t}$, and $\mathbf{U} = \mathbf{U}^{(1)} + \Delta t \mathbf{w}$.

5.2.2 Spatial discretization

When dealing with nonlinear equations, such as the Navier-Stokes equation, *linearization* is needed in order to solve the equations. In the nonlinear terms, we simply replace one (or more) of the unknown \mathbf{v} 's with a "guess" to get an

equation linear in \mathbf{v} . This guess is denoted as $\mathbf{v}^{(0)}$. Since we have three unknown functions, we use a mixed function space with three function spaces, Φ , η and Ψ . In the fluid we multiply the momentum equation with Φ , the continuity equation with η and the equation for mesh velocity with Ψ and integrate over the domain in its *current* configuration, Ω^t . In the fluid, this yields

$$\frac{\rho_f}{\Delta t}(\mathbf{v}, \Phi)_{\Omega_f} + \rho_f(((\mathbf{v} - \mathbf{w}) \cdot \nabla) \mathbf{v}^{(0)}, \Phi)_{\Omega_f^t} - (p, \nabla \cdot \Phi)_{\Omega_f^t} + 2\mu_f(\epsilon(\mathbf{v}), \nabla \Phi)_{\Omega_f^t} =$$

$$\frac{\rho_f}{\Delta t}(\mathbf{v}^{(1)}, \Phi)_{\Omega_f^t} - (\sigma_f(p, \mathbf{v}) \cdot \mathbf{n}, \Phi)_{\partial \Omega_f^t} - (\sigma_f(p, \mathbf{v}) \cdot \mathbf{n}_f, \Phi)_{\Gamma^t}$$

$$-(\nabla \cdot \mathbf{v}, \eta)_{\Omega_f^t} = 0$$

$$\begin{aligned} \Delta t(\nabla \mathbf{w}, \nabla \Psi)_{\Omega_f^t} &= -(\nabla \mathbf{U}^{(1)}, \nabla \Psi)_{\Omega_f^t} + ([\nabla \mathbf{U}^{(1)} + \Delta t \nabla \mathbf{w}] \cdot \mathbf{n}, \Psi)_{\partial \Omega_f^t} \\ &\quad + ([\nabla \mathbf{U}^{(1)} + \Delta t \nabla \mathbf{w}] \cdot \mathbf{n}_f, \Psi)_{\Gamma^t} \end{aligned}$$

And in the solid

$$\begin{aligned} \frac{\rho_s}{\Delta t}(\mathbf{v}, \Phi)_{\Omega_s} + \rho_s((\mathbf{v} \cdot \nabla) \mathbf{v}^0, \Phi) + \Delta t(\sigma_s(\mathbf{v}), \nabla(\Phi))_{\Omega_s} &= \frac{\rho_s}{\Delta t}(\mathbf{v}^{(1)}, \Phi)_{\Omega_s} \\ -(\sigma_s(\mathbf{U}^{(1)}), \nabla \Phi)_{\Omega_s} - ([\sigma_s(\mathbf{U}^{(1)}) + \Delta t \sigma_s(\mathbf{v})] \cdot \mathbf{n}, \Phi)_{\partial \Omega_s^t} \\ &\quad - ([\sigma_s(\mathbf{U}^{(1)}) + \Delta t \sigma_s(\mathbf{v})] \cdot \mathbf{n}_s, \Phi)_{\Gamma^t} \end{aligned}$$

$$\frac{1}{\delta}(\mathbf{v}, \Psi)_{\Omega_s} - \frac{1}{\delta}(\mathbf{w}, \Psi)_{\Omega_s} = 0$$

The parameter δ should be small and ensures the importance of $\mathbf{v}_s = \mathbf{w}_s$ inside the solid. On the interface, we have distinguished between the normal vector with respect to the fluid and solid domain. In general $\mathbf{n}_f = -\mathbf{n}_s$. To be able to set up and assemble the matrices for this system, the equations should be added to form one bilinear form $a(\mathbf{v}, p, \mathbf{w}, \Phi, \eta, \Psi)$ and one linear form $L(\Phi, \eta, \Psi)$.

5.2.3 A discussion on functions spaces

We have previously defined the L^2 (def. 4.1) and H^1 (def. 4.2) spaces, as well as the linear continuous galerkin basis functions (section 4.1.2). In order to have a well posed-problem, we need a triple (Φ, η, Ψ) to satisfy a few given conditions. In the following, a brief justification of the choice of function spaces used in this study are given.

For the incompressible Navier-Stokes equations, much of the mathematical theory and understanding have been developed by investigation of the simplified Stokes flow where the acceleration term is neglected in the momentum equation, that is

$$-\mu \nabla^2 \mathbf{u} + \nabla p = \mathbf{f}$$

Numerous possible pairs (Φ, η) have been proposed over the years since the first report by Taylor and Hood [43]. The discretization used by Taylor and Hood consists of quadratic piecewise polynomials for the velocity components and linear piecewise polynomials for the pressure and is still a very popular choice of basis functions. These types of elements are often referred to as Taylor-Hood elements or simply just P2-P1 elements.

As mentioned earlier, the final step in the finite element method consists of solving a linear system of equations. In the case of Stokes equations with body forces \mathbf{f} , a matrix system on the following form needs to be solved

$$\begin{bmatrix} \mathbf{A} & \mathbf{B} \\ \mathbf{B}^T & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{v} \\ \mathbf{p} \end{bmatrix} = \begin{bmatrix} \mathbf{f} \\ \mathbf{0} \end{bmatrix}$$

Which means that

$$\mathbf{A}\mathbf{v} + \mathbf{B}\mathbf{p} = \mathbf{f} \quad (5.1)$$

$$\mathbf{B}^T \mathbf{v} = \mathbf{0} \quad (5.2)$$

To get an expression for \mathbf{v} , we multiply (5.1) with \mathbf{A}^{-1} to obtain

$$\mathbf{v} = \mathbf{A}^{-1}(\mathbf{f} - \mathbf{B}\mathbf{p})$$

And insert this expression into (5.2) to get an equation only involving the pressure

$$\mathbf{B}^T \mathbf{A}^{-1}(\mathbf{f} - \mathbf{B}\mathbf{p}) = \mathbf{0}$$

or

$$\mathbf{B}^T \mathbf{A}^{-1} \mathbf{B} \mathbf{p} = \mathbf{B}^T \mathbf{A}^{-1} \mathbf{f}$$

For the solution to be unique, the matrix $\mathbf{B}^T \mathbf{A}^{-1} \mathbf{B}$ often referred to as the *Schur complement* needs to be non-singular. A necessary and sufficient condition for this is that $\text{Ker}(\mathbf{B}) = \{0\}$, or

$$\sup_{\mathbf{v}_h} \int p_h \nabla \cdot \mathbf{v}_h > 0$$

For all discrete pressures $p_h \neq 0$. This ensures solvability. For convergence, the famous Babuska-Brezzi (BB), or inf-sup condition needs to be satisfied [44]

$$\inf_{p_h} \sup_{\mathbf{v}_h} \frac{\int_{\Omega} p_h \nabla \cdot \mathbf{v}_h}{\|\mathbf{v}_h\|_1 \|p_h\|_0} \geq D > 0$$

Where D is a constant independent of the mesh resolution.

Provided this condition is satisfied the following error estimate should hold for the Stokes equations

$$\|\mathbf{v}_h - \mathbf{v}\|_1 + \|p_h - p\|_0 < C(h^k \|\mathbf{v}\|_{k+1} + h^{l+1} \|p\|_{l+1})$$

Where k and l are the degrees of polynomials used for velocity components and pressure, respectively. To obtain optimal convergence for the solution the polynomial degree should be one higher for the velocity components than for the pressure, that is, $k = l + 1$. For instance, using P3-P1 elements, computer resources are "wasted" by introducing more degrees of freedom (dofs) without improving convergence. Several choices of element type combinations, for instance Linear elements (P1) both for velocity and pressure do not satisfy the BB condition, and as a consequence unphysical oscillations in pressure can be seen.

Elements not satisfying the BB condition can be used if a proper stabilization is introduced. Due to the drastical reduction in dofs, P1-P1 elements are often used when large systems are solved for instance in commercial software. This combination is default in COMSOL, while for 3D problems in FLUENT, a slightly different element, the "mini-element" is used as first developed by Fortin [45]. This element is linear but with an extra degree of freedom known as a bubble in the center.

In this study, P2-P1 elements are used for the material velocity and pressure. However, a function space is also needed for the domain velocity, \mathbf{w} . As discussed by Quaini [46], P1-elements for the domain velocity will ensure the transformation of straight lines in the new domain. In the fluid momentum equation, the function \mathbf{w} is only used in the term $((\mathbf{v} - \mathbf{w}) \cdot \nabla) \mathbf{v}^{(0)}$ and since \mathbf{v} is a polynomial of degree 2, $\mathbf{v} - \mathbf{w}$ will also be a polynomial of degree 2. Therefore, the

5.2.4 Treatment of boundary conditions

In addition to the boundary conditions described in the original benchmark paper from Turek and Hron, homogeneous Dirichlet conditions are prescribed to the mesh displacement velocity on the domain boundary, i.e

$$\mathbf{w} = 0 \text{ on } \partial\Omega_f^t \cup \partial\Omega_s^t$$

Except for the fluid velocity on the outlet, the domain boundaries (not interface) have prescribed Dirichlet conditions on both \mathbf{u} and \mathbf{w} . Therefore the test functions Φ and Ψ will be zero on these boundaries.

If we add all the equations in the previous section together, the contributions to the boundary integral on the interface gives:

$$-(\sigma_f(p, \mathbf{v}) \cdot \mathbf{n}_f, \Phi)_{\Gamma^t} - ([\sigma_s(\mathbf{U}^{(1)}) + \Delta t \sigma_s(\mathbf{w})] \cdot \mathbf{n}_s, \Phi)_{\Gamma^t}$$

By leaving this out of the variational form, we weakly impose

$$\sigma_f(p, \mathbf{v}) \cdot \mathbf{n} = \sigma_s(\mathbf{U}) \cdot \mathbf{n}$$

On the interface. The choice of \mathbf{n} ($\mathbf{n} = \mathbf{n}_f$ or $\mathbf{n} = \mathbf{n}_s$) is arbitrary, but the same for each side of the equation.

Because we use the same function for fluid velocity and solid velocity, the no-slip condition is naturally incorporated for the fluid on the structure

$$\mathbf{v}_f = \mathbf{v}_s \text{ on } \Gamma^t$$

Because the functions \mathbf{v}_f and \mathbf{v}_s share nodes on the interface.

The additional equation for \mathbf{w} in the fluid also gives rise to boundary conditions on $\frac{\partial \mathbf{U}}{\partial n}$ on the interface. To this end we set

$$\frac{\partial \mathbf{U}}{\partial n} = 0$$

and rather let the parameter δ underline the importance of $\mathbf{w} = \mathbf{v}$ *inside* the solid, whereas \mathbf{w} in the fluid should just ensure a smooth mesh displacement.

On the outlet, we assign the stress-free condition $\sigma_f(p, \mathbf{v}) \cdot \mathbf{n} = 0$ so the boundary integral also vanish on the outlet for the momentum equation in the fluid.

This means that all integrals involving boundaries will vanish in the variational form. The Dirichlet conditions are imposed in FEniCS as previously described.

5.2.5 FSI in FEniCS

There will be some changes and a great leap in complexity compared to the previous example using FEniCS. The main differences and additions are explained here. One thing to highlight is the always ongoing changes and updates in the dolfin library. Therefore, if a solver was to be used by someone other than the writer, it will constantly need updates and fixes. This thesis do not intend to present a full solver with great complexity and many dependencies, but rather outline the most important lines of code and explain difficulties behind the FSI problem in FEniCS. The explanation here intends that a reader somewhat familiar with FEniCS should be able to implement such a code within a short amount of time.

The computational mesh is constructed in gmsh with a straight boundary dividing the fluid and the solid. This way, the class `MeshFunction` can be utilized by dividing the mesh in two subdomains. We now assume we have classes describing the solid and fluid region, implemented with functions similiar to the boundary functions in the Poisson example.

```
mesh = Mesh('FSI_mesh.xml')
SD = MeshFunction('uint', mesh, mesh.topology().dim())
SD.set_all(0)
Elastic().mark(SD,1)
```

where

```
class Elastic(SubDomain):
    def inside(self,x,on_bnd):
        # returns True if vector x in solid.
```

'uint' means that the MeshFunction has values of nonnegative integers. The last argument ensures the MeshFunction to have the same dimension as the mesh. Using the MeshFunction, the fluid domain have been marked 0, and the solid domain have been marked 1. Integration over the two domains can be separated by passing this number to `dx` in the variational formulation. A similar class, the FacetFunction

```
boundaries = FacetFunction("size_t",mesh)
```

is used to mark the boundaries and, if needed, separate integration over specific parts of the boundary.

'uint' means that the MeshFunction has values of nonnegative integers. 'size_t' means the same for the FacetFunction. The last argument to MeshFunction ensures the MeshFunction to have the same dimension as the mesh.

We need a function spaces for all three testfunctions, corresponding to `v`, `p` and `w`, and in this case we can use a handy FEniCS class to create a mixed function space. Test -and trial functions should also be created from this mixed space.

```
V = VectorFunctionSpace(mesh,'CG',2)
P = FunctionSpace(mesh,'CG',1)
W = VectorFunctionSpace(mesh,'CG', 1)
VPW = MixedFunctionSpace([V,P,W])
v,p,w = TrialFunctions(VPW)
phi,eta,psi = TestFunctions(VPW)
```

All Dirichlet boundary conditions need to be specified, and the functions need to be in the space of the respective trial function where the condition is set. For instance, the top boundary of the domain have been marked 2 with the FacetFunction, and we want to prescribe the no-slip condition on the fluid velocity.

```
noslip = Constant((0.0,0.0))
bcv2 = DirichletBC(VPW.sub(0),noslip,boundaries,2) # Top
```

All the Dirichlet boundary conditions are put together in a list, `bcs`.

When the Mesh -and FacetFunctions have been properly marked, we need to map the information from these classes to the different measures, `dx`, `ds` and `dS` representing integration over cells, exterior facets and interior facets, respectively. This is done by:

```
dS = Measure('dS')[boundaries]
dx = Measure('dx')[SD]
ds = Measure('ds')[boundaries]

dx_f = dx(0,subdomain_data=SD)
dx_s = dx(1,subdomain_data=SD)
```

The last two lines simplifies the integrands in the variational form and make it more clear which expressions are to be used in the fluid domain and which should be used in the solid domain.

Sometimes, it can be convenient to define the constants used in FEniCS as instances of the class `Constant`, to avoid re-compiling if the value of the constant is changed. e.g.

```
dt = 0.0003
k = Constant(dt)
```

We can now attention our focus to the variational form. Regular Python functions can be used in the variational formulation, and by defining these two

```
def sigma_s(U):
    return 2*mu_s*sym(grad(U)) + lamda*tr(sym(grad(U)))*Identity(2)

def eps(v):
    return sym(grad(v))
```

To be able to start the simulation some initial conditions are needed, and by setting

```
U = Function(W)
v1 = Function(V)
v0 = Function(V)
```

The initial conditions are simply set to be zero both for velocity and displacement.

The variational form is very similar to the mathematics. We define the bilinear and linear forms, a and L , for each separate equation, momentum, continuity and the movement of the domain in both the fluid and solid domain (except for continuity in the solid). For instance, a_{MF} will denote the bilinear form, a , for the momentum equation in the fluid.

```

# FLUID
aMF = rho_f/k*inner(v,phi)*dx_f \
      + rho_f*inner(grad(v0)*(v-w),phi)*dx_f \
      - inner(p,div(phi))*dx_f \
      + 2*mu_f*inner(eps(v),grad(phi))*dx_f

LMF = rho_f/k*inner(v1,phi)*dx_f

aCF = -inner(div(v),eta)*dx_f

aDF = k*inner(grad(w),grad(psi))*dx_f
LDF = -inner(grad(U),grad(psi))*dx_f

aF = aMF + aCF + aDF
LF = LMF + LDF

# SOLID
aMS = rho_s/k*inner(v,phi)*dx_s \
      + rho_s*inner(grad(v0)*v,phi)*dx_s \
      + k*inner(sigma_s(v),grad(phi))*dx_s

LMS = rho_s/k*inner(v1,phi)*dx_s \
      - inner(sigma_s(U),grad(phi))*dx_s

aDS = 1/delta*inner(v,w)*dx_s \
      - 1/delta*inner(d,w)*dx_s

aS = aMS + aDS
LS = LMS

```

We can now add the forms together to obtain one bilinear and one linear form

```

a = aS + aF
L = LS + LF

```

Before the time loop starts we define a function for holding the solution:

```

VPW_ = Function(VPW)

```

This function will consist of all values for \mathbf{v} , p and \mathbf{w} .

The time loop runs until the current time exceeds the specified end time, T . The forms change in time, and thus needs to be assembled to be updated to use the correct values for $\mathbf{v}^{(1)}$, $\mathbf{w}^{(1)}$, $\mathbf{U}^{(1)}$ and $\mathbf{v}^{(0)}$. The linear form needs an update each time step, while the bilinear form needs to be updated every single iteration inside the time loop. For the iterative method, we have chosen the Picard iteration based on the simplicity of the algorithm compared to Newton's method, especially when dealing with a mixed function space consisting of three separate spaces. (In fact, Balaban [47], wrote a thesis on Newton's method for this problem, but to the authors knowledge the algorithms have not been updated to be compatible with newer versions of Dolfin). The iteration runs until

the L^2 norm of $(\mathbf{v} - \mathbf{v}^{(0)})$ is less than a given number, τ , or if the number of iterations becomes to large.

```

while t < T:
    ...
    b = assemble(L)
    ...
    while error > tau and k_iter < max_iter:
        A = assemble(a)
        A = ident.zeros()
        [bc.apply(A,b) for bc in bcs]
        solve(A,VPW_.vector,b,'lu')
        v_,p_,w_ = VPW_.split(True)
        eps = errornorm(v_,v0,degree_rise=3)
        k_iter += 1

    v0.assign(v_)

```

The second statement within the iteration loop is needed because the lack of an equation for p within the solid. The *ident.zeros()* function replaces zeros with ones on the diagonal of the matrix block, and the solution vector for p will be zero inside the solid. In FEniCS, a Function has to be defined over the whole mesh, and adjusting the linear system as described is a way to overcome this issue in the present version of DOLFIN (1.6.0). To assign a new value for $v0$, and later be able to calculate drag and lift, we split the solution vector with the argument `True`. The solver is 'lu' by default, but in this case it is written explicitly. Iterative solvers are in general way faster, but in this case no Krylov Solver was found to converge.

The next problem to address is how the mesh should be updated. The domain should now move with velocity \mathbf{w} , so we want to move the mesh with $\Delta t \mathbf{w}$ from one time step to the next. For the total displacement, the update $\mathbf{U} = \mathbf{U}^{(1)} + \Delta t \mathbf{w}$ should also be taken into account. The actual update of the mesh is done with the functions *ALE.move()* and *bounding_box_tree().build()*

```

w_vector()[:] *= float(k)
U_.vector()[:] += w_vector()[:]
ALE.move(mesh,w_)
mesh.bounding_box_tree().build(mesh)

v1.assign(v_)

```

The final line is to update the velocity, so we can move to the next time step. Note that the velocity in both the fluid and the solid is updated by this call.

5.3 Benchmark Results

5.3.1 CFD tests

For the CFD tests we perform tests treating the flag as a rigid object. This can be done by changing the structural parameters, or simply by adjust the mesh to include the fluid domain only. In this validation we choose the latter. We show convergence with Mesh, where mesh 0 is the coarsest version. The **Ref.** are the reference values as given in the original benchmark paper.

Parameter	CFD1	CFD2	CFD3
$\rho_f [10^3 \frac{\text{kg}}{\text{m}^3}]$	1	1	1
$\nu_f [10^{-3} \frac{\text{m}^2}{\text{s}}]$	1	1	1
\bar{v}_0	0.2	1	2
$\text{Re} = \frac{Ud}{\nu_f}$	20	100	200

Table 5.1: Parameters for the CFD test cases

cells	dofs	Drag	Lift
1334	6443	13.9344	1.0980
5336	24892	14.1165	1.0836
21344	97808	14.1865	1.0944
Ref.		14.29	1.119

Table 5.2: Results for CFD1

cells	dofs	Drag	Lift
1334	6443	130.092948352	10.9117261826
5336	24892	134.43022177	10.473965217
21344	97808	135.777285175	10.7118857057
Ref.		136.7	10.53

Table 5.3: Results for CFD2

5.3.2 CSM tests

The structural tests are performed by adding the gravitational force to the structural part only. The CSM3 test is computed as a time-dependent case, starting from the initial position while CSM1 and CSM2 are Steady State (SS) solutions. For CSM3, the total energy is not conserved as the Backward Euler scheme used to discretize in time slightly recudes the amplitude for oscillating solutions. For this reason, a third parameter of interest is included in the results, namely the reduction of amplitude from one cycle to the next. From

cells	dofs	Drag	Lift
1334	6443	391.305 ± 2.039	-28.536 ± 200.149
5336	24892	428.769 ± 5.735	-18.001 ± 429.410
21344	97808	1	2
Ref.		439.45 ± 5.6183	-11.893 ± 437.81

Table 5.4: Results for CFD3 for $\Delta t = 0.0005$ and $\Delta t = 0.0001$

the reference results it appears that the flag bounces above the initial position in steady state, meaning that some energy must have been added due to their choice of scheme. This is also clear when closely examining the attached plots. This was not further discussed, and no time-dependent amplitude was reported. The temporal discretization in the original benchmark proposal was done by the Crank-Nicholson scheme, which in general have better conservation properties than the Backward-Euler scheme but is known to be less stable [48]

Parameter	CSM1	CSM2	CSM3
$\rho_s [10^3 \frac{\text{kg}}{\text{m}^3}]$	1	1	1
ν_s	0.4	0.4	0.4
$\mu_s [10^6 \frac{\text{m}^2}{\text{s}}]$	0.5	2	0.5
$g [\frac{\text{m}^2}{\text{s}}]$	2	2	2

Table 5.5: Parameters for the CSM test cases

cells	dofs	U_x of A [10^{-3}]	U_y of A [10^{-3}]
738	4596	-12.410569	-60.599246
2952	17305	-12.419505	-60.622920
11808	67077	-12.422290	-60.630433
Ref.		-7.187	-66.10

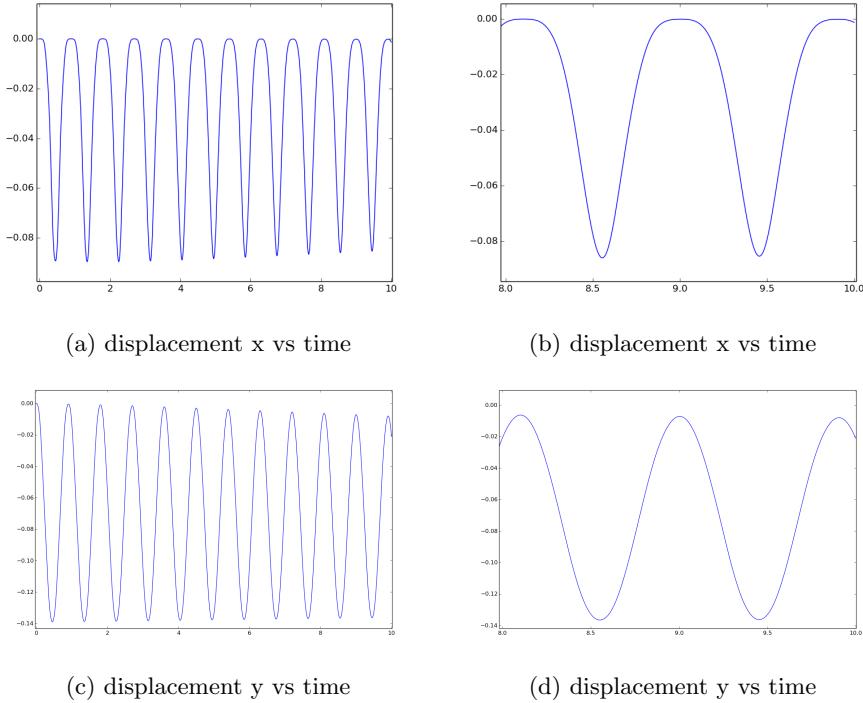
Table 5.6: Results for CSM1

cells	dofs	U_x of A [10^{-3}]	U_y of A [10^{-3}]
738	4596	-0.92479395	-16.853778
2952	17305	-0.92558954	-16.861757
11808	67077	-0.92583356	-16.864252
Ref.		-0.4690	-16.97

Table 5.7: Results for CSM2

cells	dofs	U_x of A [10^{-3}]	U_y of A [10^{-3}]	Amp reduction y (%)
738	4596	-44.175 ± 44.007	-72.712 ± 65.281	5.4
2952	17305	-44.176 ± 44.001	-72.720 ± 65.281	5.4
11808	67077	-44.177 ± 44.001	-72.720 ± 65.281	5.4
Ref.		-14.305 ± 14.305	-63.607 ± 65.160	

cells	dofs	U_x of A [10^{-3}]	U_y of A [10^{-3}]	Amp reduction y (%)
738	4596	-44.631 ± 44.628	-69.702 ± 69.290	0.3
2952	17305	-44.633 ± 44.629	-69.701 ± 69.289	0.3
11808	67077	-44.094 ± 44.091	-69.703 ± 69.289	0.3
Ref.		-14.305 ± 14.305	-63.607 ± 65.160	

Table 5.8: Results for CSM3, $\Delta t = 0.01, 0.001$ 

5.3.3 FSI tests

Parameter	FSI1	FSI2	FSI3
$\rho_f [10^3 \frac{\text{kg}}{\text{m}^3}]$	1	1	1
$\nu_f [10^{-3} \frac{\text{m}^2}{\text{s}}]$	1	1	1
\bar{v}_0	0.2	1	2
$\text{Re} = \frac{U_d}{\nu_f}$	20	100	200

Parameter	FSI1	FSI2	FSI3
$\rho_s [10^3 \frac{\text{kg}}{\text{m}^3}]$	1	10	1
ν_s	0.4	0.4	0.4
$\mu_s [10^6 \frac{\text{m}^2}{\text{s}}]$	0.5	0.5	2

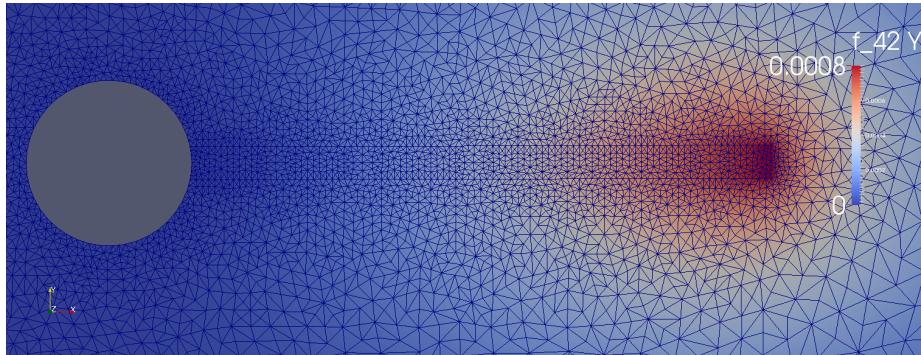


Figure 5.3: Steady State displacement in y-direction for the FSI1 test case for the medium refinement version of the mesh. Note that the mesh around the structural part also has been slightly adjusted

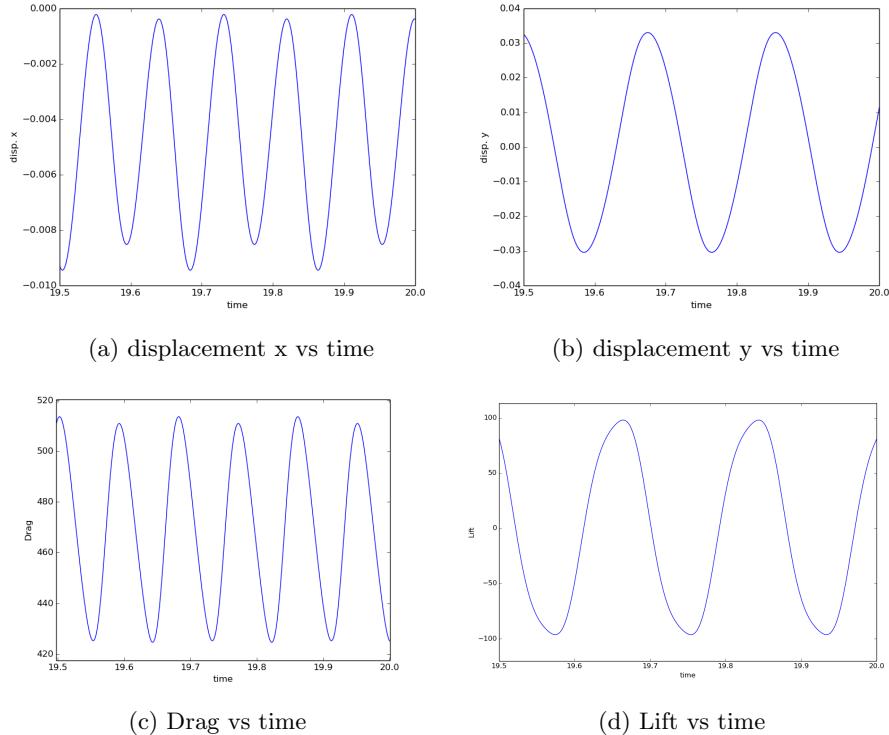
cells	dofs	U_x of A [10^{-3}]	U_y of A [10^{-3}]	Drag	Lift
2698	15329	0.015596	0.74221	14.0876279441	0.756130219216
10792	60336	0.017738	0.77686	14.1777783843	0.763145083966
43168	239384	0.019824	0.79558	14.1869409712	0.758109277348
Ref.		0.0227	0.8209	14.295	0.7638

Table 5.9: Results for FSI1

- note: Hron, Turek, $\Delta t = 0.0005$, we used $\Delta t = 0.0003$.

cells	dofs	U_x of A [10^{-3}]	U_y of A [10^{-3}]	Drag	Lift
2698	15329	-4.33 ± 4.54	1.40 ± 29.96	441.45 ± 33.15	-2.30 ± 178.00
10792	60336	-4.84 ± 4.62	1.27 ± 31.74	469.11 ± 44.50	0.92 ± 97.27
43168	239384				
Ref.		-2.69 ± 2.53	1.48 ± 34.38	457.3 ± 22.66	2.22 ± 149.78

Table 5.10: Results for FSI3



Numbers for FSI3: Ux: Max = 0.0002099, Min = -0.008879
 Uy: Max = 0.03136, Min = -0.02856
 Drag: Max = 474.6, Min = 408.3
 Lift: Max = 175.7, Min = -180.3 (174.4, -177.0)

Frequency, 0.1795 /s for medium mesh.

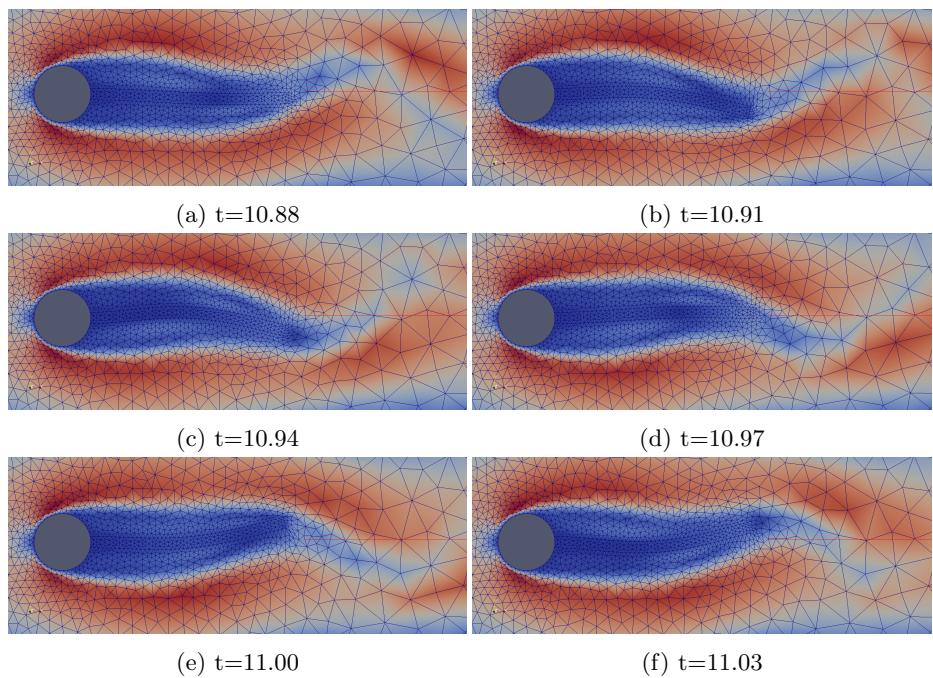


Figure 5.5: The colormap shows the magnitude of the velocity around the flag at six different states of time in fully developed flow on the coarsest mesh. Maximum velocity reaches 4.37. The mesh consists of a smooth curve at the interface, and the domains are separated beforehand in FEniCS

Chapter 6

Numerical Methods for the Biot Problem

The equations describing the Biot problem are quite similar to the elasticity case, but from the modeling point of view, some problems arise. We have previously taken advantage of the continuous Galerkin elements in the sense that the fluid velocity is continuous with the structural velocity. The boundary condition

$$\mathbf{v}_f = \mathbf{v}_s + \mathbf{q} \quad (6.1)$$

Will not allow us to have one continuous function describing fluid velocity \mathbf{v} , in the fluid domain and skeleton velocity \mathbf{v}_s , in the poroelastic medium. The right hand side of the boundary condition describes the total (macroscopic) velocity in the poroelastic medium. In the following we show a workaround for this problem, and it should now be stressed that the notation have to change in a way to implement the problem in FEniCS. We now use the following

\mathbf{v} – fluid velocity in the fluid domain. Total velocity (darcy flux + skeleton velocity) in the poroelastic domain.

\mathbf{w} – domain (or mesh) velocity in the fluid domain. Skeleton velocity in the poroelastic domain.

p – fluid pressure in the fluid domain. Pore pressure in the poroelastic domain.

\mathbf{U} – domain (or mesh) displacement in the fluid domain. Skeleton displacement in the poroelastic domain.

6.1 Weak form of the equations

The weak form is obtained in a similar way as for the FSI-problem. The notation has changed slightly and the equations in the poroelastic domain are slightly different. By omitting the inertia term $\frac{d\mathbf{q}}{dt}$, the momentum equation (3.13) for the fluid phase can be written

$$\mathbf{q} = -K\nabla p - \rho_p \frac{dw}{dt}$$

And inserting this into equation (6.1), now using \mathbf{w} as the skeleton velocity gives

$$\mathbf{v} = \mathbf{w} - K\nabla p - \rho_p \frac{dw}{dt}$$

The weak form reads:

$$\begin{aligned} \frac{\rho_f}{\Delta t}(\mathbf{v}, \Phi)_{\Omega_f} + \rho_f(((\mathbf{v} - \mathbf{w}) \cdot \nabla) \mathbf{v}^{(0)}, \Phi)_{\Omega_f^t} - (p, \nabla \cdot \Phi)_{\Omega_f^t} + 2\mu_f(\epsilon(\mathbf{v}), \nabla \Phi)_{\Omega_f^t} = \\ \frac{\rho_f}{\Delta t}(\mathbf{v}^{(1)}, \Phi)_{\Omega_f^t} - (\sigma_f(p, \mathbf{v}) \cdot \mathbf{n}, \Phi)_{\partial\Omega_f^t} - (\sigma_f(p, \mathbf{v}) \cdot \mathbf{n}_f, \Phi)_{\Gamma^t} \end{aligned}$$

$$-(\nabla \cdot \mathbf{v}, \eta)_{\Omega_f^t} = 0$$

$$\begin{aligned} \Delta t(\nabla \mathbf{w}, \nabla \Psi)_{\Omega_f^t} = -(\nabla \mathbf{U}^{(1)}, \nabla \Psi)_{\Omega_f^t} + ([\nabla \mathbf{U}^{(1)} + \Delta t \nabla \mathbf{w}] \cdot \mathbf{n}, \Psi)_{\partial\Omega_f^t} \\ + ([\nabla \mathbf{U}^{(1)} + \Delta t \nabla \mathbf{w}] \cdot \mathbf{n}_f, \Psi)_{\Gamma^t} \end{aligned}$$

And in the poroelastic medium

$$\begin{aligned} \frac{\rho_s}{\Delta t}(\mathbf{w}, \Phi)_{\Omega_s} + \rho_s((\mathbf{w} \cdot \nabla) \mathbf{w}^0, \Phi) + \Delta t(\sigma_s(\mathbf{w}), \nabla(\Phi))_{\Omega_s} = \frac{\rho_s}{\Delta t}(\mathbf{w}^{(1)}, \Phi)_{\Omega_s} \\ - (\sigma_s(\mathbf{U}^{(1)}), \nabla \Phi)_{\Omega_s} - ([\sigma_s(\mathbf{U}^{(1)}) + \Delta t \sigma_s(\mathbf{w})] \cdot \mathbf{n}, \Phi)_{\partial\Omega_s^t} \\ - ([\sigma_s(\mathbf{U}^{(1)}) + \Delta t \sigma_s(\mathbf{w})] \cdot \mathbf{n}_s, \Phi)_{\Gamma^t} \end{aligned}$$

$$\frac{1}{\delta}(\mathbf{v}, \Psi)_{\Omega_s} - \frac{1}{\delta}(\mathbf{w}, \Psi)_{\Omega_s} + \frac{K}{\delta}(\nabla p, \psi) + \rho_f \frac{K}{\delta \Delta t}(\mathbf{w}, \Psi) = \rho_f \frac{K}{\delta \Delta t}(\mathbf{w}^0, \Psi)$$

Chapter 7

Material parameters

In the simulations the units millimeters and grams are used. This combination gives back the SI-unit $\frac{N}{m^2} = \frac{kg \cdot m}{s^2 \cdot m^2} = \frac{g}{s^2 \cdot mm} = Pa$ for pressure, and is also convenient when considering the scale of the spinal cord. CSF consists of 99% water [49], and thus CSF is modeled as water at $37^\circ C$, i.e $\rho_f = 10^{-3} \frac{g}{mm^3}$ and $\nu_f = 0.658 \frac{mm^2}{s}$.

For the spinal cord, studies have shown a huge variety in material parameters. One of the most measured properties in the mammalian central nervous system is probably the Young's modulus, E according to Smith, Humphrey [50]. In addition to this, values for the Poisson ratio, ν_P needs to be found. In the literature, there is a huge gap in reported Young's modulus for spinal cord tissue. These studies does not distinguish between grey and white matter in the spinal cord, and neither will we. In general, grey and white matter will have different elastic and porous properties, however as shown by Støverud et al (2015) [11], the distinction between the two have shown to have negligible effect except for in small local regions depending on patient-specific distribution of the two. Developing patient-specific models are highly relevant for a precise description of the CSF flow, but is not the main goal for this thesis.

From this, Lame's parameters for the spinal cord were determined as

$$\mu_s = \frac{E}{2(1 + \nu_P)}$$

and

$$\lambda_s = \frac{\nu_P E}{(1 + \nu_P)(1 - 2\nu_P)} = 3.9 \cdot 10^4 \text{ Pa}$$

The spinal cord has fibres oriented in the axial direction and a direction-dependent Youngs modulus would then be expected. In the literature values range between 0.012 to 1.98 MPa. As reported by [55] most of spinal cord experimental studies use a tensile test, and the stress-strain and stress-relaxation responses of the spinal cord are non-linear. Therefore, it will in general be hard to compare results from different studies using different arbitrary levels of strain. Another approach used by Kwon [57], focuses more on spinal cord injuries and are thus more interested in properties during compression. From the modeling point of view, both approaches is of interest, and better constitutive models could be attained by combining results from several of these studies. In this work, however,

Article	Region	Model	Parameters
Hung et al. [51]	spinal cord	Experimental	$E = 0.26 \text{ MPa}$
Ben-Hatira et al. [52]	spinal cord	Nonlinear elastic	$E = 1.4 \text{ MPa}$ $\nu = 0.499$
Ozawa et al. [53]	spinal cord	Experimental	$E = 16 \text{ kPa}$
Smith and Humphrey [50]	brain	Experimental	$E = 5.0 \text{ kPa}$ $\nu = 0.479$
Cheng et al. [54]	spinal cord	Review	$E = 0.0119\text{-}1.98 \text{ MPa}$
Clarke [55]	spinal cord	Review	$E = 0.012\text{-}1.37 \text{ MPa}$
Persson et al. [56]	spinal cord	Review (linear elastic)	$E = 0.26\text{-}1.3 \text{ MPa}$

Table 7.1: Summary of elastic parameters used in literature (as presented in Klystad [10])

we limit Youngs modulus to be a constant independent of spatial direction. For the spinal cord, this assumption has been made in all the previous cited works in this thesis.

The permeability, κ is used as a measurement for the how fluid flows in a porous medium. A large permeability indicates a pervious medium. We use the value from [Karen, Ida]

$$\kappa = 1.4 \cdot 10^{-15} \text{ m}^2$$

Chapter 8

Simulating interaction between CSF and the Spinal Cord

8.1 Overview of previous studies

To our knowledge, few studies has investigated fluid-structure interaction modeling of the CSF and the spinal cord including a syrinx.

Some studies have investigated the effects of FSI on the spinal cord movement and CSF pressure in geometries without a syrinx. Clark [58] assumed the Youngs modulus to be 1 MPa for the spinal cord, and in their initial tests this choice caused only small displacements. Therefore the conclusion was that FSI had a negligible effect on CSF pressure in the SAS.

Cheng et al. [59] investigated FSI effects on a patient-specific 3D-geometry. In their model they assumed a Youngs modulus of 0.7 MPa, and reached the same conclusion as Clark. As highlighted: *"This study informs that fluid structure interaction has no effect on CSF pressure"*.

Clearly, a too high elastic modulus will undermine the effects of FSI, if they do exist. Considering the wide range of material parameters reported in the literature for the spinal cord, we believe further investigation is necessary to be able to make such a statement. In addition, these two studies does not seem to investigate syringomelia as a primary target, and therefore important effects of FSI could have been overlooked in these specific cases. It has correctly been argued that the pressures involved in these studies ($< 100Pa$) are almost negligible in magnitude compared to even the lowest estimates of the elastic modulus of the Spinal cord ($> 5kPa$). This should cause only very small displacements, and this also seem to be the case for healthy subjects. However, when the syrinx is large, there is only a thin membrane of spinal cord tissue separating the SAS and the fluid in the syrinx allowing larger cord movements.

In porous models presented by Drøsdal, [26] fluid pressure within the cord was

altered by the presence of a syrinx but the CSF pressure in the SAS was not. velocities up to only 3e-7 cm/s inside the syrinx was reported, and therefore there must be some other effects causing the more rapid fluid movement inside the syrinx.

To our knowledge, the most noted group working on FSI effects on Syringomelia include the group of Chris D. Bertram at the University of Sydney. Some of their work include research on pressure waves propagating in the spinal cord in the presence of a syrinx [60],[61],[9]. Even though, in this series of papers the main focus is on the overlap between the cervical and thoracic segments of the spinal cord, the mechanisms of interest remains the same.

Even with today's high quality magnetic resonance imaging, (MRI) or phase contrast MR (PC-MR), exact velocities are hard to measure. Healthy subjects also has very complex CSF flow and thus difficult to observe or quantify. Since the Chiari I malformation is associated with abnormal CSF flow, a realistic model simulating the pre-operative case needs abnormal inflow or pressure boundary conditions. The latter is extremely hard to measure exact.

8.1.1 Pressure measurements in patients with Chiari I

Williams (1981) [62] and Häck (2001) [63] investigated the pressure gradient between the intracranial and spinal (lumbar) CSF compartments. In their work the pulsatile pressure were not analyzed, which seem to be more reliable in predicting intracranial compliance according to more recent studies by Eide [64, 65, 66, 67]. Frič and Eide also measured the pulsatile pressure gradient between the two compartments and have found significantly higher (mean wave amplitude, MWA) gradients in patients with evidence of syringomyelia (12/26 patients) than in those without. (3.7 ± 2.0 mmHg vs 2.1 ± 1.3 mmHg; $p = 0.02$) [68]. See figure 8.1 for a graphical representation.

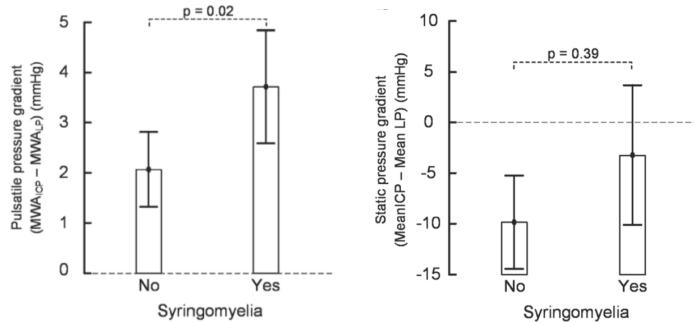


Figure 8.1: Pulsatile (left) and static pressure gradient in patients as used in the study by Frič and Eide [68]

The pulsatile intracranial pressure (ICP) as well as pulsatile pressure gradients were clearly abnormal or with borderline values in 69 and 71 % of Chiari I patients, respectively. Without any further speculation, it is interesting to note

that these numbers are very close to the number of Chiari I patients that develops a syrinx. The median pressure difference in patients with abnormal pressure gradient were 2.6 mm Hg between the intracranial CSF pressure and the lumbar (LP) CSF pressure. The actual mean static ICP was measured to a median of 7.1 mmHg for the patients (range -0.7 – 13.0), whereas the median of the mean lumbar pressure LP in the patients were 15.1 mm Hg. Czosnyka and Pickard [69] reported ICP for healthy adult subjects to be 7 – 15 mm Hg. Williams [70] measured pressure up to 70 mm Hg and 97 mmHg in the SAS in the cisternal (just below the cerebellum) and lumbar region, respectively during coughing. Sansur et al. [71] measured the SAS pressure at the L5-level and found pressures up to 125 mm Hg in patients coughing. The baseline pressures for healthy subjects were reported to be 8.6 – 13.4 mm Hg.

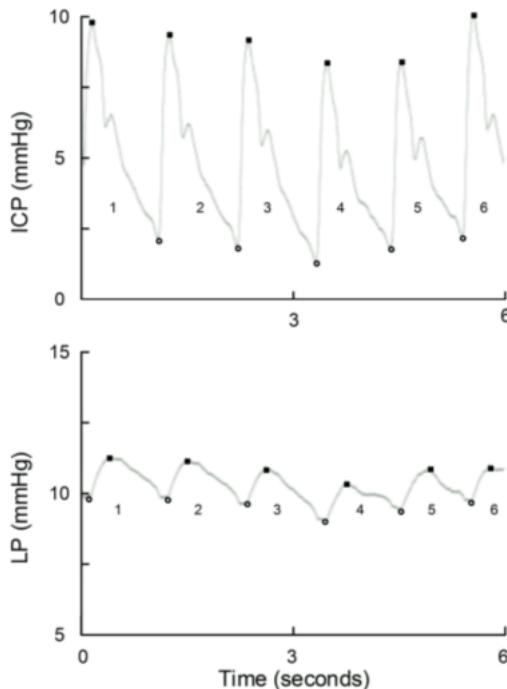


Figure 8.2: Intracranial (ICP) and lumbar (LP) pressure measured simultaneously by Frič and Eide

8.1.2 Applicability of medical measurements

From the modeling point of view, pressure measurements are of high interest in prescribing the correct boundary conditions at the top and bottom of the cord and SAS model. There are some challenges however. From our understanding, the main focus in most of these studies seem to be to measure pressure differences in Chiari patients *compared* to healthy subjects. As showed by all studies

in the previous section, altered pressure environment, and thus altered CSF dynamics is somehow related to syringomyelia. However, from a CFD point of view, the actual values of the pressure is also of high interest, and in particular the pressure difference between the Intracranial region and the lumbar region for a specific patient. Simultaneous measurements of ICP and LP was measured for the first time by Frič and Eide [68]. They measured the pressure over night in subjects laying down. The mean LP was found to have a median value 8mmHg higher than the mean ICP. In addition to this, the pressure was always higher at the Lumbar region as depicted in figure 8.2. From the fluid mechanics point of view, this implies flow in the cranial direction at all times since no hydrostatic pressure should be present when laying down. This is in contradiction to results obtained by Haughton/Bruker (not published? (Brucker ASSR 2015.pptx)) where the net flux seem to be in the caudal direction. As for now, these measurements can probably be used for prescribing a normalized pressure waveform at the inlet, but to the authors opinion the actual pressure measurements can not to be used as boundary conditions to obtain reliable results.

(Comment on experiments by Martin et al. [72])?

8.2 CSF velocities in syringomyelia

Substantial flow within the syrinx have been reported, e.g. by Brugières et al. (2000) [73] where large syrinxes (graded A or AB) had mean peak velocities of 2.93 cm/s, and small syrinxes (graded B or C) had mean peak velocities of 1.5 cm/s. Haughton/Brucker reported flow up to 3.1 cm/s inside the syrinx in an assessment of CSF velocities and Cord Motion Before and After Chiari 1 Decompression. (Raw data available xxx). Fluid flow within the syrinx is also supported by Pinna et al. (2000) [74], and it was also pointed out that flow direction inside the syrinx did not necessarily parallel with those observed in the SAS, and that these patterns may vary from patient to patient.

We hypothesized that FSI-effects (deformation and pressure wave propagation) was at least partially responsible for the syrinx velocities reported in the literature.

xxx ALSO READ 5, 13, 21, 45 from Karen xxx

In Brucker/Haughton CSF flow was measured with PCMR on three different stages: Pre-operative, 2 months post-operative and 10 months post-operative when the patient had no remaining symptoms. The peak velocity was reported to be 9.4 cm/s in the caudal direction for the pre-operative case. The velocities varied a lot over a cross-section of the cord, meaning the peak velocity 9.4 cm/s is not representative for the general velocity in the SAS. For the Cross section depicted in figure 8.4 the maximal velocities were registered at the upper left and upper right part of the circle. Along the axis of the cord, velocities in these areas lies around 4-6 cm/s.

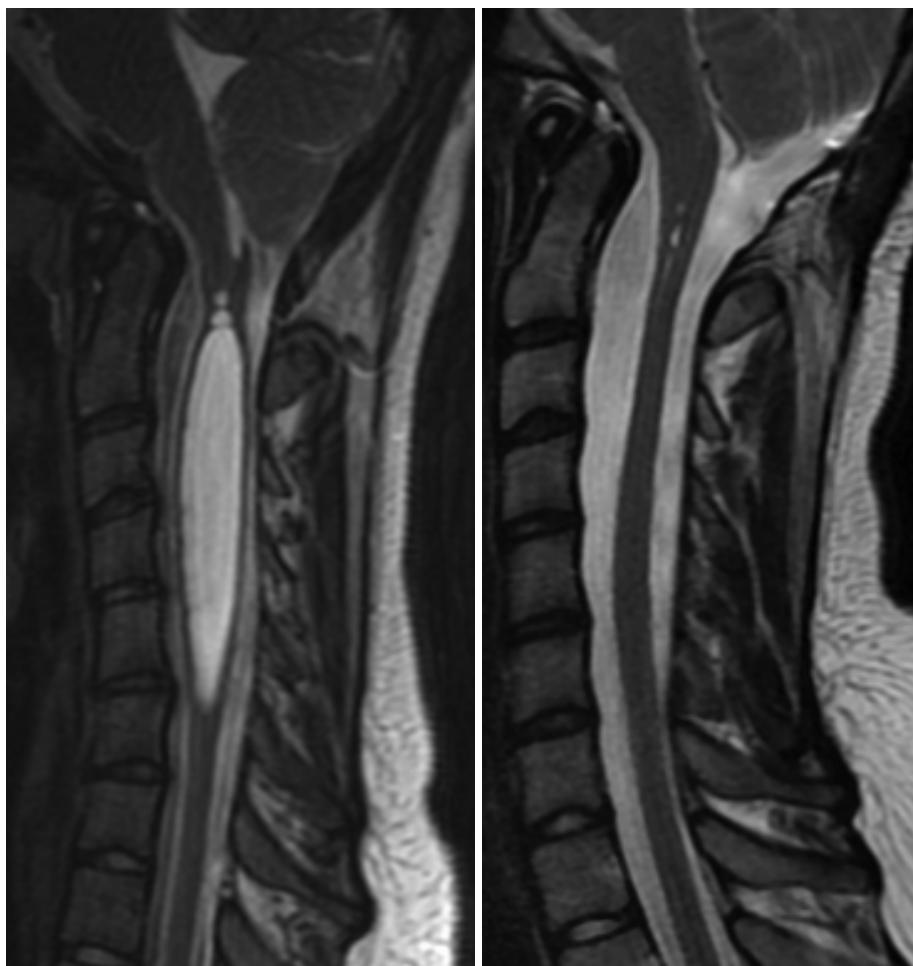


Figure 8.3: MRI image of 14 year old female subject before and 10 months after decompression. Note the withdrawal of the cerebellar tonsils in the post-operative image on the right

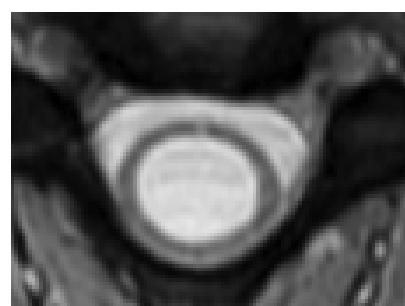
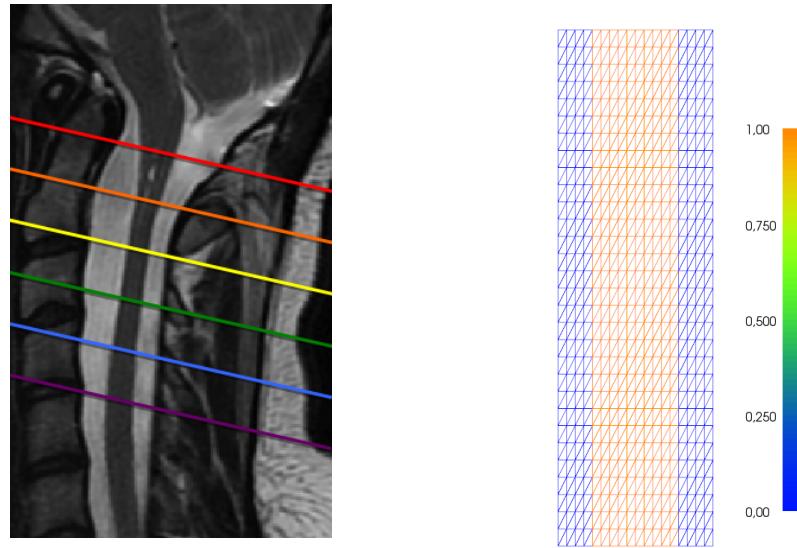
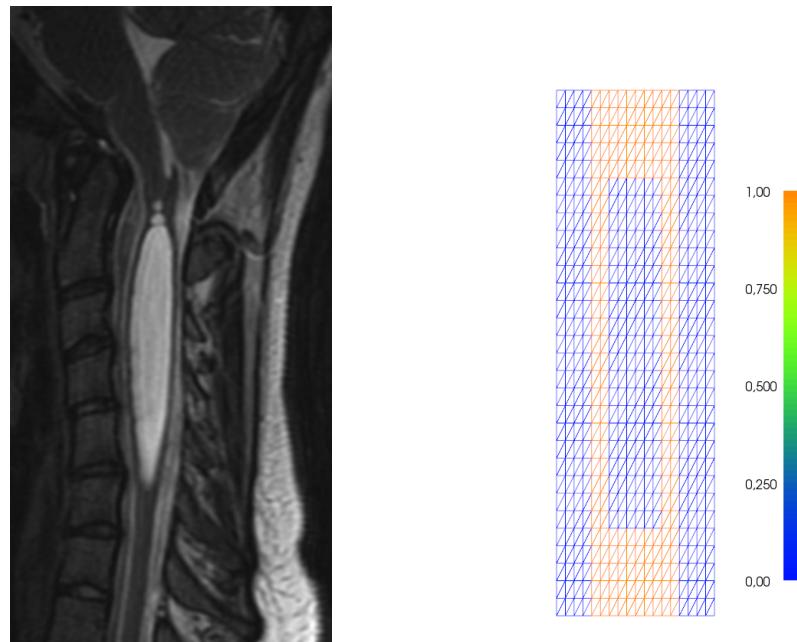


Figure 8.4: Cross-section of the spinal cord. The bottom of this cross-section is towards the face and the top is towards to the neck



(a) Levels (from top to bottom) C1-C2, C2, C2-C3, C3, C3-C4 and C4-C5 of the spinal cord

(b) Coarse version of the computational mesh



(a) Cord with fluid filled syrinx

(b) Computational mesh with a fluid filled cavity

Figure 8.6: Comparison of the spinal cord an the coarsest versions of the computaitonal meshes

8.3 A note on interface conditions

Since we want continuity of stresses on the interface, we use the divergence form (3.6) of the momentum equation, to obtain these when integrating by parts. For the outflow and inflow boundaries for the fluid the pseudo traction condition are the condition of interest. The unknown shear term $\mathbf{n} \cdot (\nabla \mathbf{v}^T)$ should then be added to the bilinear form in the variational formulation. I.e with the pseudo traction condition we have

$$\begin{aligned} \cdot \mathbf{n} = -p\mathbf{n} + \mu(\nabla \mathbf{v} + (\nabla \mathbf{v})^T) \cdot \mathbf{n} &= [-p\mathbf{n} + \mu\mathbf{n} \cdot \nabla \mathbf{v}] + \mathbf{n} \cdot (\nabla \mathbf{v}^T) \\ &= [-p\mathbf{n} + \mu \frac{\partial \mathbf{v}}{\partial n}] + \mathbf{n} \cdot (\nabla \mathbf{v}^T) \\ &= -p_0 \mathbf{n} + \mathbf{n} \cdot (\nabla \mathbf{v}^T) \quad (8.1) \end{aligned}$$

It should be noted that the *grad* function in FEniCS is in fact given such that $\text{grad}(\mathbf{v}) = (\nabla \mathbf{v})^T$, and the extra term in (8.1) can be written

```
grad(v) . T*n
```

in FEniCS. So far we have only dealt with symmetric operations regarding this function and therefore this distinction did not have to be made before this point. Similarly, the term

$$\frac{\partial \mathbf{U}}{\partial n}$$

can in fact be included in the variational form by adding the term

```
k*inner(grad(w('-'))*n('-'),psi('-'))*dS(5)
```

to the domain bilinear form aDF in section 5.2.5. And

```
inner(grad(U('-'))*n('-'),psi('-'))*dS(5)
```

to the linear form LDF in the same set of equations. The interface have been given the value 5 and ('-') is used to distinguish variables in the fluid domain from the solid domain. This should give no restriction on \mathbf{U} except that it will be continuous over the interface. These terms were omitted in the validation of the solver. The inclusion of these terms gave a small effect on displacements without a syrinx, but close to no effect on syrinx velocity or displacements in the presence of a syrinx. Deviations in x-displacements from the benchmark-case could arise by the fact that these terms were left out in that section, but this is something that will not be further investigated at the present time, and we keep these terms in the following simulations.

8.4 Results, elastic cord

The previously described models based on a elastic and poroelastic cord is used on a geometry describing idealized versions of the spinal cord. The meshes have

the same dimensions as found in previous work by Drøsda (2011), [26]. The height of the model is 60mm, and the spinal cord have a radius of 5mm. The distance from the cord to the dura mater is 4 mm, and this space is where CSF flows. The fluid cavity, where free fluid flow is allowed, is placed in the centre of the model, extending between heights 10mm and 50mm in the longitudinal direction. We investigate three cases: No syrinx, a case with syrinx 1 mm radius and a syrinx with 3 mm radius.

First, a sinusodial varying pressure was applied to the boundaries with a maximum of 20 Pa difference between top and bottom. This applied pressure gradient was reported by Drøsda to result in peak velocities of around 5-6 cm/s assuming a rigid, but porous cord. The Simulations were run 10 cycles where a steady state was reached. Quantities of interest are the peak velocity in the SAS, $\max|\mathbf{v}|$, the peak velocity inside the syrinx in the spinal canal, $\max|\mathbf{v}_{sc}|$ and the maximum displacement $\max|\mathbf{U}|$ in both x- and y-direction.

8.4.1 No syrinx

N_x	N_y	dofs	$\max \mathbf{v} $ [cm/s]	$\max \mathbf{U}_x $ [mm]	$\max \mathbf{U}_y $ [mm]
18	30	6281	5.68	0.003	0.02
36	60	24437	5.63	0.003	0.02
54	90	54473	5.63	0.003	0.02

Table 8.1: $E = 5kPa, \Delta t = 0.002s, T = 10, \rho_s = 1.75\rho_f$

N_x	N_y	dofs	$\max \mathbf{v} $ [cm/s]	$\max \mathbf{U}_x $ [mm]	$\max \mathbf{U}_y $ [mm]
18	30	6281	5.68	0.001	0.006
36	60	24437	5.62	0.001	0.006
54	90	54473	5.62	0.001	0.006

Table 8.2: $E = 16kPa, \Delta t = 0.002s, T = 10, \rho_s = 1.75\rho_f$

N_x	N_y	dofs	$\max \mathbf{v} $ [cm/s]	$\max \mathbf{U}_x $ [mm]	$\max \mathbf{U}_y $ [mm]
18	30	6281	5.67	3e-4	0.002
36	60	24437	5.61	3e-4	0.002
54	90	54473	5.62	3e-4	0.002

Table 8.3: $E = 62.5kPa, \Delta t = 0.002s, T = 10, \rho_s = 1.75\rho_f$

note: max displacement in x occurs at approx y=55 without syrinx. (y=40 w syrinx)

8.4.2 1mm Syrinx

N_x	N_y	dofs	$\max \mathbf{v} [\text{cm/s}]$	$\max \mathbf{v}_{sc} [\text{cm/s}]$	$\max \mathbf{U}_x [\text{mm}]$	$\max \mathbf{U}_y [\text{mm}]$
18	30	6281	5.94	2.10	0.20	0.08
36	60	24437	5.91	2.14	0.20	0.08
54	90	54473	5.91	2.17	0.20	0.08
72	120	96389	5.92	2.18	0.20	0.08

Table 8.4: $E = 5kPa, \Delta t = 0.002s, T = 10s, \rho_s = 1.75\rho_f$

N_x	N_y	dofs	$\max \mathbf{v} [\text{cm/s}]$	$\max \mathbf{v}_{sc} [\text{cm/s}]$	$\max \mathbf{U}_x [\text{mm}]$	$\max \mathbf{U}_y [\text{mm}]$
18	30	6281	5.76	0.57	0.05	0.02
36	60	24437	5.68	0.57	0.05	0.02
54	90	54473	5.69	0.58	0.05	0.02
72	120	96389	5.70	0.58	0.05	0.02

Table 8.5: $E = 16kPa, \Delta t = 0.002s, T = 10s, \rho_s = 1.75\rho_f$

N_x	N_y	dofs	$\max \mathbf{v} [\text{cm/s}]$	$\max \mathbf{v}_{sc} [\text{cm/s}]$	$\max \mathbf{U}_x [\text{mm}]$	$\max \mathbf{U}_y [\text{mm}]$
18	30	6281	5.69	0.14	0.01	0.005
36	60	24437	5.63	0.14	0.01	0.005
54	90	54473	5.64	0.14	0.01	0.005
72	120	96389	5.64	0.14	0.01	0.005

Table 8.6: $E = 62.5kPa, \Delta t = 0.002s, T = 10s, \rho_s = 1.75\rho_f$

Discuss results

8.4.3 3mm Syrinx

N_x	N_y	dofs	$\max \mathbf{v} [\text{cm/s}]$	$\max \mathbf{v}_{sc} [\text{cm/s}]$	$\max \mathbf{U}_x [\text{mm}]$	$\max \mathbf{U}_y [\text{mm}]$
18	30	6281	7.62	2.34	1.30	0.24
36	60	24437	7.15	2.54	1.13	0.23
54	90	54473	7.05	2.61	1.10	0.22
72	120	96389	7.01	2.78	1.07	0.22

Table 8.7: $E = 5kPa, \Delta t = 0.002s, T = 10s, \rho_s = 1.75\rho_f$

N_x	N_y	dofs	$\max \mathbf{v} [\text{cm/s}]$	$\max \mathbf{v}_{sc} [\text{cm/s}]$	$\max \mathbf{U}_x [\text{mm}]$	$\max \mathbf{U}_y [\text{mm}]$
18	30	6281	6.11	0.79	0.34	0.06
36	60	24437	6.01	0.83	0.33	0.06
54	90	54473	6.00	0.84	0.33	0.06
72	120	96389	6.00	0.85	0.33	0.06

Table 8.8: $E = 16\text{kPa}, \Delta t = 0.002\text{s}, T = 10\text{s}, \rho_s = 1.75\rho_f$

N_x	N_y	dofs	$\max \mathbf{v} [\text{cm/s}]$	$\max \mathbf{v}_{sc} [\text{cm/s}]$	$\max \mathbf{U}_x [\text{mm}]$	$\max \mathbf{U}_y [\text{mm}]$
18	30	6281	5.79	0.21	0.07	0.01
36	60	24437	5.70	0.21	0.07	0.01
54	90	54473	5.71	0.21	0.07	0.01
72	120	96389				

Table 8.9: $E = 62.5\text{kPa}, \Delta t = 0.002\text{s}, T = 10\text{s}, \rho_s = 1.75\rho_f$ Figure 8.7: Pressure fields at $t=9.35$ with $E = 5\text{kPa}$ for a thin (radius 1mm) and thick (radius 3mm) syrinx. The pressure vector in the elastic cord have been set to 0. The pressure field inside the syrinx arises due to cord displacements. The pressure gradient is in the opposite direction inside the syrinx causing flow in the opposite direction as compared to the outside SAS.

8.5 Results, poroelastic cord

max \mathbf{v} [cm/s]	max \mathbf{U}_x [mm]	max \mathbf{U}_y [mm]	max \mathbf{q} [cm/s]	max \mathbf{q}_0 [cm/s]
5.67	2e-4	0.002	1.06e-7	3.4e-9

Table 8.10: $E = 62.5 \text{ kPa}$, $\Delta t = 0.002 \text{ s}$, $T = 10$, $\rho_s = 1.75 \rho_f$, $[N_x, N_y] = [18, 30]$

max \mathbf{v} [cm/s]	max \mathbf{U}_x [mm]	max \mathbf{U}_y [mm]	max \mathbf{q} [cm/s]	max \mathbf{q}_0 [cm/s]
5.62	2e-4	0.002	2.42e-7	3.4e-9

Table 8.11: $E = 62.5 \text{ kPa}$, $\Delta t = 0.002 \text{ s}$, $T = 10$, $\rho_s = 1.75 \rho_f$, $[N_x, N_y] = [54, 90]$

8.6 The effect of asymmetric pressure

As mentioned in section 8.1.2, pressure measurements comparing the top and bottom of the cord is hard to obtain. If the temporal data in figure 8.2 can be trusted the temporal variation in the lumbar region is almost negligible compared to the wave amplitude in the cranial region. For this reason, we use the ICP measurements as pressure difference between the top and bottom of the model and scale this data to obtain approximately expected CSF velocities. However, it should be noted that pressure measurements were done in the lumbar region, lower than the bottom part of the computational model presented here, and therefore the wave amplitude would not have been damped as much as down at the lumbar region. This is by no means perfect, but in light of theories describing formations of a syrinx, it models a more realistic case than prescribing symmetric pressure.

The applied pressure was obtained by extracting raw data from ICP measurements over one representative cycle (1.16 s) and fitting to a 5-th degree spline. The spline does not capture all oscillations but is reasonable within the limitations described above. The pressure were set to 0 on the bottom of the cord. Both boundary conditions were set with the Pseudo-traction condition previously described.

8.7 Discussion

Our experiments have given further evidence that FSI-effects on CSF-dynamics are negligible models of healthy subjects. A 10-fold increase in Youngs modulus gave only a decrease in 0.01 cm/s for these models, and no visible effect on CSF-dynamics. Cord displacements vary approximately linear with Youngs modulus which would be expected with a linear elasticity model, and the greatest displacements were only $3 \mu\text{m}$ and $20 \mu\text{m}$ in the radial and axial directions, respectively.

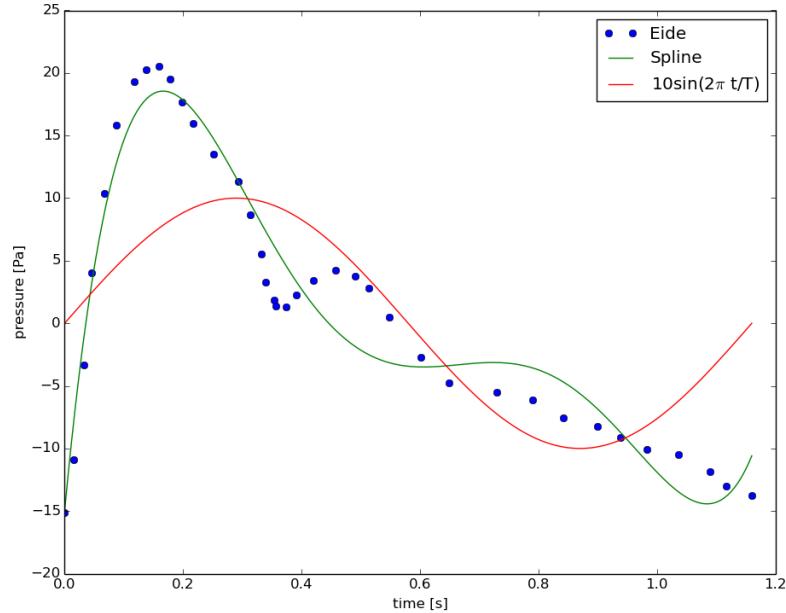


Figure 8.8: Pressure difference between top and bottom of the cord in comparison to measured ICP by Eide. In order to compare pressure characteristics, the sine function have been stretched out to match the period $T=1.16$ from the measured data. xxx DECIDE magnitude of Spline xxx

The presence of a syrinx, leaves only a small segment of spinal cord tissue between it and the surrounding SAS. This allows for greater displacements of the cord and thus fluid movement inside the syrinx.

8.8 Limitations

Kent Evju, turbulence.
 Martin2002, Eide pressure, CSF production, sinusoidal
 Discuss Eide in light of Bertram reflections.
 Effect of Pia mater

Bibliography

- [1] K. S. Paul, R. H. Lye, F. A. Strang, and J. Dutton, “Arnold-chiari malformation: review of 71 cases,” *Journal of neurosurgery*, vol. 58, no. 2, pp. 183–187, 1983.
- [2] N. Di Lorenzo, L. Palma, E. Paletinsky, and A. Fortuna, ““conservative” cranio-cervical decompression in the treatment of syringomyelia-chiari i complex: A prospective study of 20 adult cases.,” *Spine*, vol. 20, no. 23, pp. 2479–2482, 1995.
- [3] F. Guo, M. Wang, J. Long, H. Wang, H. Sun, B. Yang, and L. Song, “Surgical management of chiari malformation: analysis of 128 cases,” *Pediatric neurosurgery*, vol. 43, no. 5, pp. 375–381, 2007.
- [4] M. F. Quigley, B. Iskandar, M. A. Quigley, M. Nicosia, and V. Haughton, “Cerebrospinal fluid flow in foramen magnum: temporal and spatial patterns at mr imaging in volunteers and in patients with chiari i malformation 1,” *Radiology*, vol. 232, no. 1, pp. 229–236, 2004.
- [5] V. M. Haughton, F. R. Korosec, J. E. Medow, M. T. Dolar, and B. J. Iskandar, “Peak systolic and diastolic CSF velocity in the foramen magnum in adult patients with chiari i malformations and in normal control participants,” *American journal of neuroradiology*, vol. 24, no. 2, pp. 169–176, 2003.
- [6] S. Linge, V. Haughton, A. E. Løvgren, K.-A. Mardal, A. Helgeland, and H. P. Langtangen, “Effect of tonsillar herniation on cyclic CSF flow studied with computational flow analysis,” *American Journal of Neuroradiology*, vol. 32, no. 8, pp. 1474–1481, 2011.
- [7] S. O. Linge, K.-A. Mardal, A. Helgeland, J. D. Heiss, and V. Haughton, “Effect of cranivertebral decompression on CSF dynamics in chiari malformation type i studied with computational fluid dynamics: Laboratory investigation,” *Journal of neurosurgery. Spine*, vol. 21, no. 4, p. 559, 2014.
- [8] B. A. Martin, R. Labuda, T. J. Royston, J. N. Oshinski, B. Iskandar, and F. Loth, “Spinal subarachnoid space pressure measurements in an in vitro spinal stenosis model: implications on syringomyelia theories,” *Journal of biomechanical engineering*, vol. 132, no. 11, p. 111007, 2010.
- [9] C. Bertram, “A numerical investigation of waves propagating in the spinal cord and subarachnoid space in the presence of a syrinx,” *Journal of Fluids and Structures*, vol. 25, no. 7, pp. 1189–1205, 2009.

- [10] N. K. Kylstad, “Simulating the viscoelastic response of the spinal cord,” 2014.
- [11] K. H. Støverud, M. Alnæs, H. P. Langtangen, V. Haughton, and K.-A. Mardal, “Poro-elastic modeling of syringomyelia—a systematic study of the effects of pia mater, central canal, median fissure, white and gray matter on pressure wave propagation and fluid movement within the cervical spinal cord,” *Computer methods in biomechanics and biomedical engineering*, pp. 1–13, 2015.
- [12] D. Van Wynsberghe, C. R. Noback, and R. Carola, *Human anatomy and physiology*. McGraw-Hill College, 1995.
- [13] A. Thompson, N. Madan, J. Hesselink, G. Weinstein, A. M. del Rio, and V. Haughton, “The cervical spinal canal tapers differently in patients with chiari i with and without syringomyelia,” *American Journal of Neuroradiology*, 2015.
- [14] B. Williams, “On the pathogenesis of syringomyelia: a review..,” *Journal of the Royal Society of Medicine*, vol. 73, no. 11, p. 798, 1980.
- [15] V. J. Katz, “The history of stokes’ theorem,” *Mathematics Magazine*, vol. 52, no. 3, pp. 146–156, 1979.
- [16] O. Reynolds, *Papers on Mechanical and Physical Subjects, The Sub-Mechanics of the Universe*, vol. 3. Cambridge University Press, Cambridge, 1903.
- [17] F. White, *Viscous Fluid Flow*. McGraw-Hill series in mechanical engineering, McGraw-Hill, 3 ed., 2006.
- [18] B. Gjevik, “Viskøse vesker og elastiske stoffer,” *Forelesninger i MEK2200*, 2002.
- [19] J. N. Newman, *Marine Hydrodynamics*. The MIT Press, 1977.
- [20] P. K. Kundu, I. M. Cohen, and D. R. Dowling, *Fluid Mechanics*. Elsevier, Academic Press, 5 ed., 2012.
- [21] H. F. Wang, *Theory of linear poroelasticity*. 2000.
- [22] D. A. Nield and A. Bejan, *Convection in Porous Media*. Springer, 4 ed., 2013.
- [23] M. A. Biot, “General theory of three-dimensional consolidation,” *Journal of applied physics*, vol. 12, no. 2, pp. 155–164, 1941.
- [24] M. A. Biot, “Theory of elasticity and consolidation for a porous anisotropic solid,” *Journal of Applied Physics*, vol. 26, no. 2, pp. 182–185, 1955.
- [25] M. Biot, “Theory of finite deformations of porous solids,” *Indiana University Mathematics Journal*, vol. 21, no. 7, pp. 597–620, 1972.
- [26] I. N. Drøsdal, “Porous and viscous modeling of cerebrospinal fluid flow in the spinal canal associated with syringomyelia,” 2011.

- [27] J. Donea, A. Huerta, J.-P. Ponthot, and A. Rodriguez-Ferran, *Encyclopedia of Computational Mechanics Vol. 1: Fundamentals., Chapter 14: Arbitrary Lagrangian-Eulerian Methods*. Wiley & Sons, 2004.
- [28] A. M. Winslow, ““equipotential” zoning of two-dimensional meshes,” tech. rep., California Univ., Livermore (USA). Lawrence Livermore Lab., 1963.
- [29] G. S. Beavers and D. D. Joseph, “Boundary conditions at a naturally permeable wall,” *Journal of fluid mechanics*, vol. 30, no. 01, pp. 197–207, 1967.
- [30] P. G. Saffman, “On the boundary condition at the surface of a porous medium,” *Studies in Applied Mathematics*, vol. 50, no. 2, pp. 93–101, 1971.
- [31] I. Jones, “Low reynolds number flow past a porous spherical shell,” in *Mathematical Proceedings of the Cambridge Philosophical Society*, vol. 73, pp. 231–238, Cambridge Univ Press, 1973.
- [32] H. P. Langtangen, *Finite Element Method*. 2015.
- [33] S. Brenner and R. Scott, *The mathematical theory of finite element methods*, vol. 15. Springer Science & Business Media, 2007.
- [34] B. Rivière, *Discontinuous Galerkin methods for solving elliptic and parabolic equations: theory and implementation*. Society for Industrial and Applied Mathematics, 2008.
- [35] B. Rivière and I. Yotov, “Locally conservative coupling of stokes and darcy flows,” *SIAM Journal on Numerical Analysis*, vol. 42, no. 5, pp. 1959–1977, 2005.
- [36] W. E. Langlois and M. O. Deville, *Slow viscous flow*. Springer, 1964.
- [37] H. C. Elman, D. J. Silvester, and A. J. Wathen, *Finite elements and fast iterative solvers: with applications in incompressible fluid dynamics*. Oxford University Press (UK), 2014.
- [38] W. L. Barth and G. F. Carey, “On a boundary condition for pressure-driven laminar flow of incompressible fluids,” *International journal for numerical methods in fluids*, vol. 54, no. 11, pp. 1313–1325, 2007.
- [39] M. Schäfer, S. Turek, F. Durst, E. Krause, and R. Rannacher, *Benchmark computations of laminar flow around a cylinder*. Springer, 1996.
- [40] S. Turek and J. Hron, *Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow*. Springer, 2006.
- [41] A. Y. Tang and N. Amin, “Some numerical approaches to solve fluid structure interaction problems in blood flow,” in *Abstract and Applied Analysis*, vol. 2014, Hindawi Publishing Corporation, 2014.
- [42] K. Selim, “Adaptive finite element methods for fluidstructure interaction and incompressible flow,” 2011.

- [43] C. Taylor and P. Hood, “A numerical solution of the navier-stokes equations using the finite element technique,” *Computers & Fluids*, vol. 1, no. 1, pp. 73–100, 1973.
- [44] F. Brezzi and M. Fortin, *Mixed and hybrid finite element methods*, vol. 15. Springer Science & Business Media, 2012.
- [45] M. Fortin, “Old and new finite elements for incompressible flows,” *International Journal for numerical methods in fluids*, vol. 1, no. 4, pp. 347–364, 1981.
- [46] A. Quaini, “Algorithms for fluid-structure interaction problems arising in hemodynamics,” 2009.
- [47] G. Balaban, “A newtons method finite element algorithm for fluid-structure interaction,” 2012.
- [48] J. Hron and S. Turek, *A monolithic FEM/multigrid solver for an ALE formulation of fluid-structure interaction with applications in biomechanics*. Springer, 2006.
- [49] R. A. Fishman, *Cerebrospinal fluid in diseases of the nervous system*. WB Saunders company, 1992.
- [50] J. H. Smith and J. A. Humphrey, “Interstitial transport and transvascular fluid exchange during infusion into brain and tumor tissue,” *Microvascular research*, vol. 73, no. 1, pp. 58–73, 2007.
- [51] T.-K. Hung, G.-L. Chang, H.-S. Lin, F. R. Walter, and L. Bunegin, “Stress-strain relationship of the spinal cord of anesthetized cats,” *Journal of biomechanics*, vol. 14, no. 4, pp. 269–276, 1981.
- [52] F. Ben-Hatira, K. Saidane, and A. Mrabet, “A finite element modeling of the human lumbar unit including the spinal cord,” *Journal of Biomedical Science and Engineering*, vol. 5, no. 3, p. 150, 2012.
- [53] H. Ozawa, T. Matsumoto, T. Ohashi, M. Sato, and S. Kokubun, “Mechanical properties and function of the spinal pia mater,” *Journal of Neurosurgery*, vol. 1, no. 1, pp. 122–127, 2004.
- [54] S. Cheng, E. C. Clarke, and L. E. Bilston, “Rheological properties of the tissues of the central nervous system: a review,” *Medical engineering & physics*, vol. 30, no. 10, pp. 1318–1337, 2008.
- [55] E. C. Clarke, “Spinal cord mechanical properties,” in *Neural Tissue Biomechanics*, pp. 25–40, Springer, 2010.
- [56] C. Persson, J. L. Summers, and R. M. Hall, “Modelling of spinal cord biomechanics: In vitro and computational approaches,” in *Neural Tissue Biomechanics*, pp. 181–201, Springer, 2010.
- [57] B. K. Kwon, T. R. Oxland, and W. Tetzlaff, “Animal models used in spinal cord regeneration research,” *Spine*, vol. 27, no. 14, pp. 1504–1510, 2002.

- [58] E. C. Clarke, D. F. Fletcher, M. A. Stoodley, and L. E. Bilston, “Computational fluid dynamics modelling of cerebrospinal fluid pressure in chiari malformation and syringomyelia,” *Journal of biomechanics*, vol. 46, no. 11, pp. 1801–1809, 2013.
- [59] S. Cheng, D. Fletcher, S. Hemley, M. Stoodley, and L. Bilston, “Effects of fluid structure interaction in a three dimensional model of the spinal subarachnoid space,” *Journal of biomechanics*, vol. 47, no. 11, pp. 2826–2830, 2014.
- [60] C. Bertram, A. Brodbelt, and M. Stoodley, “The origins of syringomyelia: numerical models of fluid/structure interactions in the spinal cord,” *Journal of biomechanical engineering*, vol. 127, no. 7, pp. 1099–1109, 2005.
- [61] C. Bertram, L. Bilston, and M. Stoodley, “Tensile radial stress in the spinal cord related to arachnoiditis or tethering: a numerical model,” *Medical & biological engineering & computing*, vol. 46, no. 7, pp. 701–707, 2008.
- [62] B. Williams, “Simultaneous cerebral and spinal fluid pressure recordings,” *Acta neurochirurgica*, vol. 59, no. 1-2, pp. 123–142, 1981.
- [63] M. Häckel, V. Beneš, and M. Mohapl, “Simultaneous cerebral and spinal fluid pressure recordings in surgical indications of the chiari malformation without myelodysplasia,” *Acta neurochirurgica*, vol. 143, no. 9, pp. 909–918, 2001.
- [64] P. K. Eide, G. Bentsen, A. G. Sorteberg, P. B. Marthinsen, A. Stubhaug, and W. Sorteberg, “A randomized and blinded single-center trial comparing the effect of intracranial pressure and intracranial pressure wave amplitude-guided intensive care management on early clinical state and 12-month outcome in patients with aneurysmal subarachnoid hemorrhage,” *Neurosurgery*, vol. 69, no. 5, pp. 1105–1115, 2011.
- [65] P. K. Eide and A. Brean, “Cerebrospinal fluid pulse pressure amplitude during lumbar infusion in idiopathic normal pressure hydrocephalus can predict response to shunting,” *Cerebrospinal Fluid Res*, vol. 7, no. 5, 2010.
- [66] P. K. Eide and E. Kerty, “Static and pulsatile intracranial pressure in idiopathic intracranial hypertension,” *Clinical neurology and neurosurgery*, vol. 113, no. 2, pp. 123–128, 2011.
- [67] P. K. Eide and W. Sorteberg, “Diagnostic intracranial pressure monitoring and surgical management in idiopathic normal pressure hydrocephalus: A 6-year review of 214 patients,” *Neurosurgery*, vol. 66, no. 1, pp. 80–91, 2010.
- [68] R. Frič and P. K. Eide, “Comparison of pulsatile and static pressures within the intracranial and lumbar compartments in patients with chiari malformation type 1: a prospective observational study,” *Acta neurochirurgica*, vol. 157, no. 8, pp. 1411–1423, 2015.
- [69] M. Czosnyka and J. D. Pickard, “Monitoring and interpretation of intracranial pressure,” *Journal of Neurology, Neurosurgery & Psychiatry*, vol. 75, no. 6, pp. 813–821, 2004.

- [70] B. Williams, "Cerebrospinal fluid pressure changes in response to coughing.," *Brain: a journal of neurology*, vol. 99, no. 2, pp. 331–346, 1976.
- [71] C. A. Sansur, J. D. Heiss, H. L. DeVroom, E. Eskioglu, R. Ennis, and E. H. Oldfield, "Pathophysiology of headache associated with cough in patients with chiari i malformation," *Journal of neurosurgery*, vol. 98, no. 3, pp. 453–458, 2003.
- [72] B. A. Martin, R. Labuda, T. J. Royston, J. N. Oshinski, B. Iskandar, and F. Loth, "Pathological biomechanics of cerebrospinal fluid pressure in syringomyelia: Fluid structure interaction of an in vitro coaxial elastic tube system," pp. 941–942, 2009.
- [73] P. Brugières, I. Idy-Peretti, C. Iffenecker, F. Parker, O. Jolivet, M. Hurth, A. Gaston, and J. Bittoun, "CSF flow measurement in syringomyelia," *American journal of neuroradiology*, vol. 21, no. 10, pp. 1785–1792, 2000.
- [74] G. Pinna, F. Alessandrini, A. Alfieri, M. Rossi, and A. Bricolo, "Cerebrospinal fluid flow dynamics study in chiari i malformation: implications for syrinx formation," *Neurosurgical focus*, vol. 8, no. 3, pp. 1–8, 2000.