

Université catholique de Louvain
Ecole Polytechnique de Louvain

Modelling of electromagnetic and piezoelectric peristaltic micropumps

Guillaume Beckers
Electromechanical engineer

PhD thesis
27th February 2018

Members of the jury

Pr.	L. Delannay	UCL - iMMC	President
Pr.	B. Dehez	UCL - iMMC	Supervisor
Pr.	P. Chatelain	UCL - iMMC	
Pr.	L. Francis	UCL - ICTM	
Pr.	Y. Perriard	EPFL (Switzerland)	
Pr.	Y. Bernard	Polytech Paris-Sud (France)	

Abstract

Micropumps are devices that can handle microlitre-scale fluid volumes. Various pumping principles have been proposed in the literature and are often coupled to valves in order to ensure a positive mean flow. In this thesis, two original designs of peristaltic valveless micropumps are introduced. The fluid is moved by the peristaltic motion of a planar diaphragm which is bent either by the action of Lorentz force, in the electromagnetic micropump, or by piezoelectric elements, in the piezoelectric micropump. The reverse flow is avoided by ensuring the contact between the diaphragm and the micropump's bottom.

Several characteristic parameters of these devices have to be determined with accuracy: their flow rate, and the back pressure they can sustain ; equally important are the electromechanical limits, i.e. the maximum stress and electrical field in order to prevent any damage to the pump. Both micropump designs present similarities in their geometries and working principle, meaning they can be studied based on the same tool. This thesis has been dedicated to the development of an efficient tool to study devices where thin diaphragms, possibly multilayered, enter in contact with an obstacle. Special attention has been paid to the modelling of displacement, stress and electric field, using both micropumps as benchmark application cases.

In the first part of this thesis, a theory is built considering a slender structure composed of several layers, using Euler-Bernoulli beam assumptions and quasistatic condition. The actions of the piezoelectric effect and of the transverse Lorentz force density, due to the interaction of electrical current density and a magnetic field, are taken into account through equivalent loads. The effect of the electrical field variation with respect to the thickness coordinate of the piezoelectric layers on the displacement is neglected but its impact on the stress is considered. Contact is taken into account by imposing three boundary conditions at the ends of the beam, considering its position as an unknown. The theory is applied to the micropump designs and compared to a 3D finite element model. This has shown that boundary and contact conditions in the width direction are not properly modelled, highlighting the need

for a more sophisticated theory.

This is developed in the second part of the thesis, using Kirchhoff-Love assumptions for thin plates. The electrical field is solved exactly and used to obtain a theory where the displacement components are the only unknowns. A full electromechanical coupling is therefore considered through equivalent loads and stiffness coefficients; dynamics are also taken into account. The contact condition is however quite difficult to handle in the differential form, so a variational form is used instead and solved by a finite element method, coupled to an active set strategy for contact management. This theory compares well with the 3D finite element model and the efficiency of the developed theories and algorithms, in terms of computational cost and memory requirements, opens the way to use it in a design process with short turn-around times or which requires a huge number of evaluations, such as in optimization process.

Acknowledgements

- C'est une bonne situation ça thésard ?

- Vous savez, moi je ne crois pas qu'il y ait de bonne ou de mauvaise situation. Moi, si je devais résumer ma vie aujourd'hui avec vous, je dirais que c'est d'abord des rencontres. Des gens qui m'ont tendu la main, peut-être à un moment où je ne pouvais pas, où j'étais seul chez moi. Et c'est assez curieux de se dire que les hasards, les rencontres forment une destinée... Parce que quand on a le goût de la chose, quand on a le goût de la chose bien faite, le beau geste, parfois on ne trouve pas l'interlocuteur en face je dirais, le miroir qui vous aide à avancer. Alors ça n'est pas mon cas [...] ce goût donc qui m'a poussé à entreprendre une thèse, mais demain qui sait ? Peut-être simplement à me mettre au service de la communauté, à faire le don, le don de soi...

reprise de Nexusis, dans le film "Astérix et Obélix"

Ce texte est l'aboutissement de six années de travail. La thèse est semée de moments éprouvants, certains exaltants, d'autres moins. Bien souvent présenté comme un chemin solitaire, je n'en serais pas là aujourd'hui sans l'aide et le soutien de plusieurs personnes que je tiens à remercier.

Tout d'abord, j'aimerais remercier mon promoteur, le professeur Bruno Dehez, pour sa patience, sa disponibilité et ses conseils avisés. Son enthousiasme contagieux, ainsi que la passion pour la recherche qu'il transmet à ses chercheurs, m'ont aidé à avancer dans ce travail de longue haleine.

J'aimerais également remercier les membres de mon jury, pour leurs apports à cette thèse, et plus spécifiquement:

- Le professeur Yves Bernard, pour m'avoir accompagné tout au long de cette thèse et pour m'avoir accueilli à deux reprises au laboratoire de génie électrique de Paris.

- Le professeur Philippe Chatelain, membre de mon comité d'accompagnement, toujours présent pour répondre à mes questions sur les écoulements de fluide.
- Les professeurs Yves Perriard et Laurent Francis, pour avoir accepté de lire ma thèse. Ainsi que le professeur Laurent Delannay qui a assuré le rôle de président de jury.

Je tiens aussi à remercier mes collègues de bureau, Geoffrey, Virginia, Corentin, Joachim, Christophe et Guillaume mais aussi Caroline, Léna, Timothée, Xavier, Maxence, Aubain, Gabriel et Sophie qui ont fait de ce passage au CEREM une expérience inoubliable, tant durant les heures de bureau qu'en dehors.

Je tiens aussi à remercier ma famille et mes amis pour leur soutien inconditionnel au cours des dernières années. Une attention toute particulière à ma fiancée, Maureen, qui m'a supporté, a résisté à mes sautes d'humeur et a écouté mes monologues, parfois incompréhensibles, sur ces modèles...

Contents

Abstract	i
Acknowledgements	iii
Contents	v
1 Introduction	1
1.1 State of the art in micropumps	2
1.1.1 Pumping principle	2
1.1.2 Flow rectification	13
1.2 New micropump concepts	15
1.2.1 Electromagnetic micropump	15
1.2.2 Piezoelectric micropump	18
1.2.3 Comparison of concepts	20
1.3 Thesis objectives	22
1.4 Research contributions	22
1.5 Thesis outline	23
I Modelling - One dimensional approach	25
2 Constitutive equations for laminated Euler-Bernoulli beams	27
2.1 Overview of literature	27
2.2 Bending of composite beams	28
2.2.1 Null ϵ_y -strain assumption	32
2.3 Electric field study	33
2.4 Lorentz loads	33
2.5 Case study - the piezoelectric bender	34
2.5.1 2D model	34
2.5.2 Numerical results	34

2.6	Conclusion	37
3	Application to micropump concepts	39
3.1	Electromagnetic micropump	39
3.1.1	Magnetic field modelling	40
3.1.2	Mechanical modelling	44
3.1.3	Finite element verification	46
3.2	Piezoelectric micropump	47
3.2.1	Finite element verification	50
3.2.2	Limits of the one dimensional modelling	53
3.3	Conclusion	58
II	Modelling - Two dimensional approach	61
4	Constitutive equations for laminated Kirchhoff-Love plates	65
4.1	Overview of literature	65
4.2	Dynamics of thin laminated plates	68
4.2.1	Dynamic equilibrium	69
4.2.2	Stress-strain relations	71
4.2.3	Electrical field study	72
4.3	Summary of the set of equations	75
4.3.1	Strong form	75
4.3.2	Weak form	79
4.4	Discretization	81
4.4.1	Spatial discretization	81
4.4.2	Time discretization	82
4.5	Numerical study	83
4.5.1	Actuator	84
4.5.2	Energy harvester	85
4.5.3	Study of stresses and electric field	87
4.6	Conclusion	88
5	Treatment of frictionless contacts	91
5.1	Geometrical and equilibrium aspects	92
5.2	Overview of literature	94
5.2.1	Formalisms and methods	95
5.2.2	Discretization	98
5.3	Description of the adopted method and results	99
5.3.1	Primal-dual active set strategy	99
5.4	Numerical Study	101
5.5	Conclusion	103

6	Application to micropump concepts	105
6.1	Electromagnetic micropump	105
6.1.1	3D model	105
6.1.2	Laminated plate model	110
6.1.3	Numerical results	111
6.2	Piezoelectric micropump	114
6.2.1	3D model	114
6.2.2	Numerical results	118
6.3	Conclusion	124
7	Conclusion and perspectives	127
7.1	Original contributions	129
7.2	Outlook	130
A	Materials' data	133
B	Finite element - technical and computational aspects	137
B.1	The mesh	138
B.2	The element	140
B.3	Local contribution and matrix assembly	143
B.4	Post-processing	147
B.5	Laminated plates	147
	Bibliography	151

Introduction

1

With the advances in biotechnologies, a new field, known as microfluidics, has emerged. It is the science and technology of systems for handling and processing amount of fluid in the range of microlitre and even smaller. This field has been elected as one of the ten '*Emerging Technologies That Will Change the World*' by the MIT Technology Review as explained in Zacks (2003). As reported in Whitesides (2006), it finds its origins in molecular analysis for the development of analytical methods, biodefense, for detection of chemical and biological substances that could present a terrorist threat, molecular biology, where high sensitivity and high resolution devices are needed for DNA sequencing. In addition, production techniques used in microelectronics helped to develop the first microfluidic chips.

Those devices are composed of various functional blocks, depending on the application. Micropump is one of these and permits to move fluid through the chip. If there is no clear definition of what covers the term micropump, in literature, it refers to a pump with at least one dimension in the micrometer range which is capable of handling small amounts of fluid in the order of the microlitre and even smaller. A direct consequence of this miniaturization is the development of new pumping principles. These are linked to phenomena which become exploitable due to the change in length scale. Depending on the application, the requirements in terms of flow rate, back pressure, volume, precision, power consumption can be quite different. This explains the tremendous amount of papers which can be found on the subject in the last 30 years. Here are some examples as reported in Laser and Santiago (2004):

- Micro total analysis systems (μ TAS), which are systems performing chemical analysis on small fluid samples and integrated on a chip. These applications can require the manipulation of samples as small as 1 pl.
- Thermal management, where an electronic chip is cooled by a fluid. It may requires flow rates of 10 ml min^{-1} and pressures of 100 kPa or higher.

- Implantable drug delivery systems, which do not require high flow rate but a precise metering of the delivered drug. The encountered back pressure can be as high as 25 kPa, which is not negligible.

In this chapter, a first section is dedicated to a review of existing designs, classified according to their pumping principle, their actuation and the rectification used to ensure a non zero mean flow rate. Then, the second section introduces the two original micropump concepts that will be the benchmark cases for the theories developed across this thesis. The objectives of this thesis will be stated in the third section. Some research questions come out from these objectives, which are given in the fourth section. This chapter ends with the thesis outlines.

1.1 State of the art in micropumps

The aim of this section is not to give an exhaustive list of all micropumps presented in the literature. Several reviewing papers have already been published, gathering several hundreds of papers on the subject, see Laser and Santiago (2004), Iverson and Garimella (2008), Yokota (2014), Woias (2005), Tsai and Sue (2007), Zhang et al. (2007), Nisar et al. (2008), Amirouche et al. (2009).

In view of the works referenced in these articles, a micropump can be described based on the physical principle used for pumping and on the flow rectification used. The goal of this section is to give a general glance at the micropump structure and especially to their pumping principle. Some additional designs of small pumps, which are not in the sub-millimetre range, are also given. Most of the information gathered in this section comes from Laser and Santiago (2004), Iverson and Garimella (2008) and Yokota (2014).

1.1.1 Pumping principle

The miniaturization of pumping devices has led to new pumping principles. Indeed, many physical phenomena, which are neglected or impossible to exploit when considering fluid flow at macroscale, can have a significant impact on microflows. Conversely, diminishing the characteristic lengths of the devices also means changing the flow characteristics, which means that macro pumps designs are not applicable anymore.

The pumping principles can be divided in two families: the first gathers the displacement pumps and the second one, the dynamic pumps. In displacement pumps, a moving part directly exerts a pressure on the fluid. In dynamic pumps, the device continuously gives energy to the fluid by increasing its momentum or its pressure.

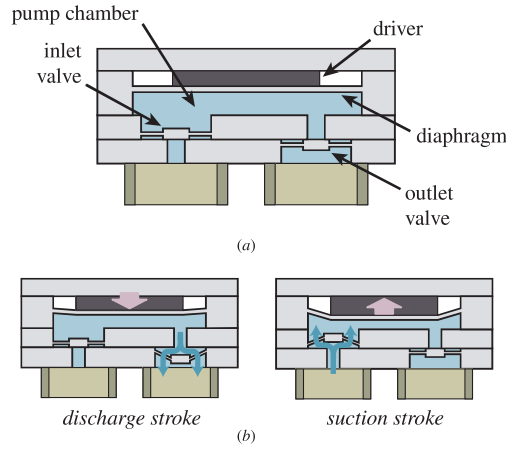


Fig. 1.1: Diaphragm displacement pump. The driver moves up the diaphragm to take fluid at the inlet and moves it down to expel it at the outlet. Fluid is rectified using valves. Image from Laser and Santiago (2004).

According to Laser and Santiago (2004), most of reported micropumps are reciprocating displacement pumps based on a moving plate, also called diaphragm, such as shown in fig. 1.1. These pumps are also called membrane pumps. They used a periodic movement of their diaphragm to change the volume of their chamber. This basically works as a piston pump, where fluid is sucked up during the increase of the volume of its chamber and expelled with its decrease. These pumps need additional components to ensure a non-zero flow rate. By putting several chambers in series, the rectification can be done intrinsically such as in the pump developed by Smits (1990) and shown in fig. 1.2. Such a pump is also called a peristaltic pump because the fluid is moved from one chamber to another, mimicking the movement of the alimentary canal. Many micropumps have been developed based on this idea. They differentiate by the actuator used to move it, as well as the way a positive mean flow is obtained.

For actuation, piezoelectric materials transforms directly electrical energy to mechanical energy. This allows very compact design with high frequency driving capabilities. Two different actuation principles are mainly used, depending on the coupling coefficient and mechanical bonding of the piezoelectric actuator. The first one uses the d_{33} coupling, with the actuator bonded to the housing on one side and to the diaphragm on the other. The actuator acts in the manner of a piston under the action of an electric field applied on the third axis i.e. the polarization axis. This type of actuator is referred to as the stack actuator. The other mode uses the d_{31} coupling, where the piezoelectric material is bonded to the diaphragm on one side and free on the other. An electric field applied along the polarization axis makes the material expand or shrink in the perpendicular plane. This creates a bending motion of the

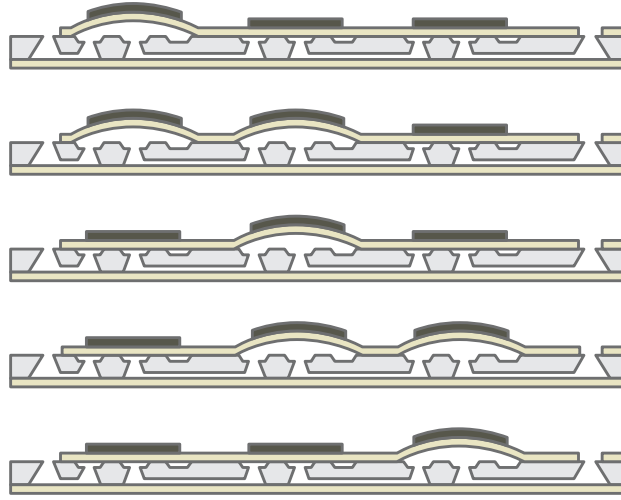


Fig. 1.2: Diaphragm displacement micropump with three chambers in series, working peristaltically. Actuation is performed by piezoelectric materials. Rectification is made by the chambers themselves and the powering scheme ensuring all chambers are not actuated at the same time. Image from Laser and Santiago (2004).

diaphragm. This type of actuation generally gives higher displacement amplitude at the expense of a lower driving force compared to the stack type. Both designs are shown in 1.3. Both of them needs additional considerations to rectify the flow.

Another possible actuation of the diaphragm is through an electrostatic force, due to the voltage difference between two parallel electrodes. One of them is fixed to the external frame of the pump and the other to the diaphragm. This actuation is based on Coulomb forces appearing between the charges on the electrodes. An example of such a design is shown in fig. 1.4. The pressure applied on the diaphragm induces a bending, which will make fluid flow in the chamber of the pump. The released of the electrostatic pressure expels the fluid out. This type of pump is generally coupled to a valve.

A third actuation principle is based on Lorentz forces appearing between a current and a magnetic field. One example is based on the same principle as the voice coil, where a magnet is tightened to the diaphragm and a current in the coil moves the diaphragm up and down. One of its advantages is the rather simple power electronics needed for its control. Some other attempts for more integrated design have been reported in Iverson and Garimella (2008). Another example is the one of Ala'aldien et al. (2015), shown in fig. 1.5, where a permanent magnet is moved in a circular tube by imposing the right current pattern in windings. A maximum flow rate of 8 ml min^{-1} and a maximum pressure of 575 Pa for two different configurations have

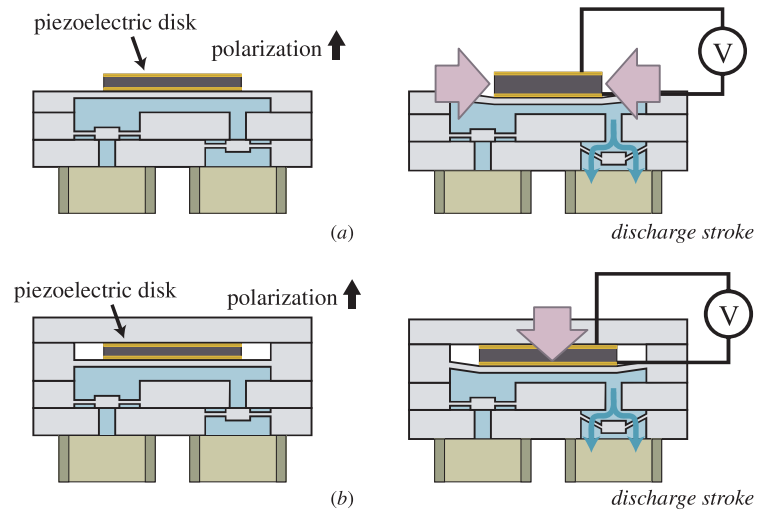


Fig. 1.3: Piezoelectric actuation of the diaphragm using (a) d_{31} coupling coefficient for bending actuation and (b) d_{33} coupling for piston like actuation. Image from Laser and Santiago (2004).

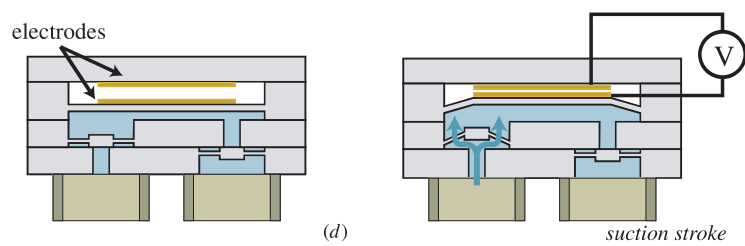


Fig. 1.4: Electrostatic actuation of the diaphragm. An applied voltage moves up the diaphragm during the suction stage. The discharge is obtained by removing this voltage, using the elastic energy stored in the diaphragm. Image from Laser and Santiago (2004).

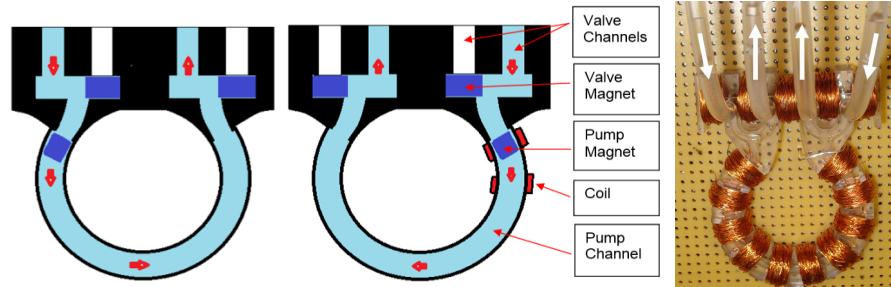


Fig. 1.5: Fluid is pumped by moving magnets in a small tube using Lorentz force between the magnet and the coils around the tubes. Rectification is done using magnets too. Several configurations have been tested. A maximum flow rate of 8 ml min^{-1} and a maximum pressure of 575 Pa for two different configurations are reported. Ala'aldeen et al. (2015)

been reported.

A fourth way to displace the diaphragm is based on pneumatic. In this design, there is a second chamber in which pressurized air is injected such as shown in fig. 1.6 (b). Another one is based on the thermal expansion of a small encapsulated air volume. The air is then heated by an electrical heater which raised the pressure in the chamber and deflect the diaphragm. This is shown in fig. 1.6 (a). Some other actuation principles based on heating have been reported. It includes the bending of a diaphragm made in two layers of materials with different thermal expansion coefficients. By heating the plate, one of the layers expands more than the other and the diaphragm bends. Another is based on the bistable behaviour encountered in shape memory alloys. By heating and cooling the alloy above and under its critical temperature, the material moves the diaphragm, creating a pumping effect.

Another type of displacement pump is the one where a second fluid is used to displace the first one, without any diaphragm. These pumps are called fluid displacement pumps. Since both fluids are in contact, they have to be immiscible for the pump to work properly. If they are miscible, an idea is to use a third fluid to separate the actuating fluid and the pumped fluid. An example of such a pump is the one reported in Yamahata et al. (2005), using a ferrofluid and an oil plug to separate. The ferrofluid is used as a piston, moved by a magnet and the fluid flow is rectified by check valves. The principle and the 3D model are shown in fig. 1.7. Some other examples are the ones where gas bubbles are used to displace the fluid. In this case, either the expansion, or contraction, of a fluid is used to displace another. The gas can be from an external source, from a phase transition or from an electrochemi-

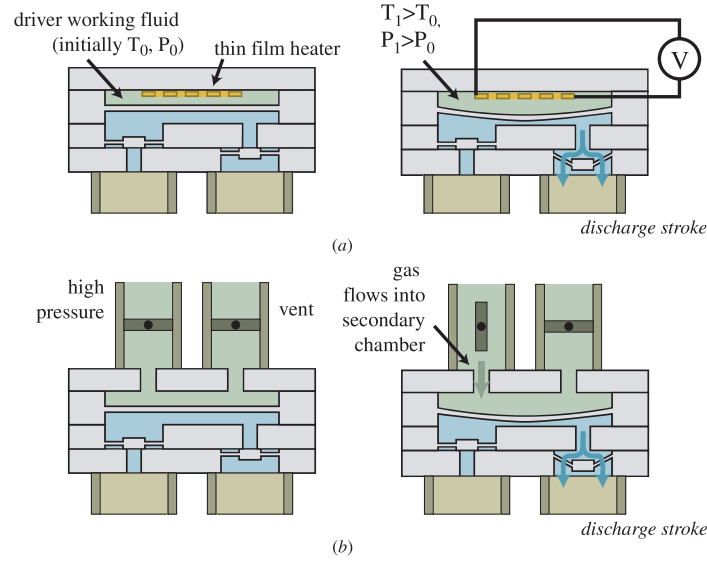


Fig. 1.6: Pneumatic actuation of the diaphragm (a) by using the change in volume of the air in the chamber above the diaphragm when it is heated and cooled or (b) by using an external source of compressed air. Image from Laser and Santiago (2004).

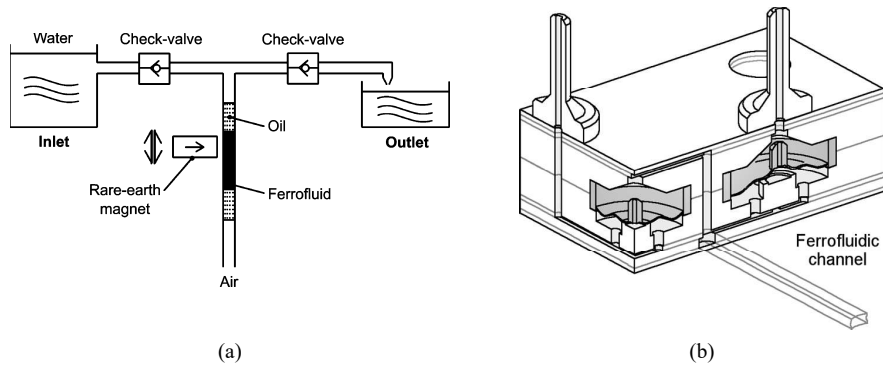


Fig. 1.7: Ferrofluid piston pump driven by a magnet studied in Yamahata et al. (2005). Principle (a) and 3D model (b). The prototype produced a flow rate of $30 \mu\text{l min}^{-1}$. The maximum back pressure is of 2.5 kPa.

cal reaction such as electrolysis. More examples are given in Iverson and Garimella (2008).

Some miniaturized and integrated version of gear pumps have also been developed, such as the one of Waldschik and Büttgenbach (2010b) shown in fig. 1.8. The pump is driven by a synchronous micro motor, which is integrated in the pump. This reduces the pump size and, in addition, the need of a sealing for the driving shaft is

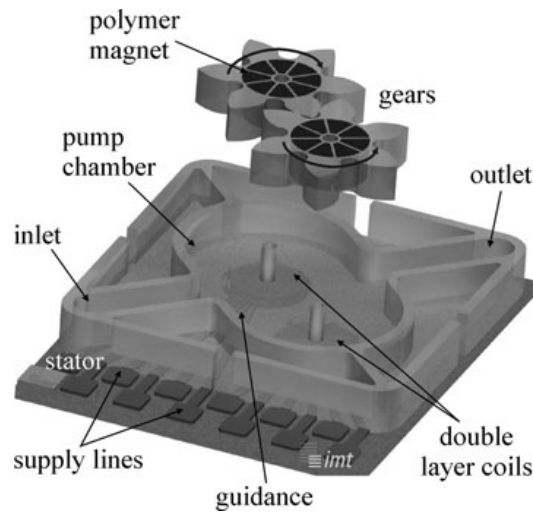


Fig. 1.8: Micro gear pump with internal electromagnetic drive. The produced prototype amounts 10 mm by 12 mm by 1.3 mm for a measured flow rate of $150 \mu\text{l}/\text{min}$ at 150 rpm. Polymer magnets have been integrated in the gears which allows a compact design without the need for a sealing of the driving shaft Waldschik and Büttgenbach (2010b)

circumvented. The produced prototype amounts 10 mm by 12 mm by 1.3 mm, for a measured flow rate of $150 \mu\text{l}/\text{min}$ at 150 rpm. No measurement or evaluation of the expected back pressure were reported in the article.

Displacement micropumps can also take advantage of dynamical effects, such as in the piezoelectric resonantly driven micropump, where the actuation frequency is matched on the resonant frequency of the system given by the bellow and the mass. This leads to a pump of 9 mm by 10 mm with a maximum flow rate of 4.8 ml min^{-1} , a maximum pumping pressure of 320 kPa and a maximum power of 8.7 mW, as reported in Park et al. (1999). The pump is shown in fig. 1.9.

The pumping principles in dynamic pumps are more diversified. Some of them are miniaturization of traditional pumps, such as the centrifugal pumps. Two examples of such pumps are shown in fig. 1.10 and 1.11. They both integrate a synchronous micro motor as actuator. If no measurements were reported in Waldschik and Büttgenbach (2010a) for the first pump, the second pump studied in Matar et al. (2017) was found to develop a maximum flow rate of $14.3 \text{ ml}/\text{min}$ at 11 400 rpm and no back pressure, as well as a maximum pressure of 1.57 kPa at 9000 rpm, for a package size in the centimetre range. Another example is the one of Nakamoto et al. (2011) in which an axial flow pump has been fabricated using micromilling tools. A maximum flow rate around $80 \text{ ml}/\text{min}$ and a pressure of 780 Pa have been

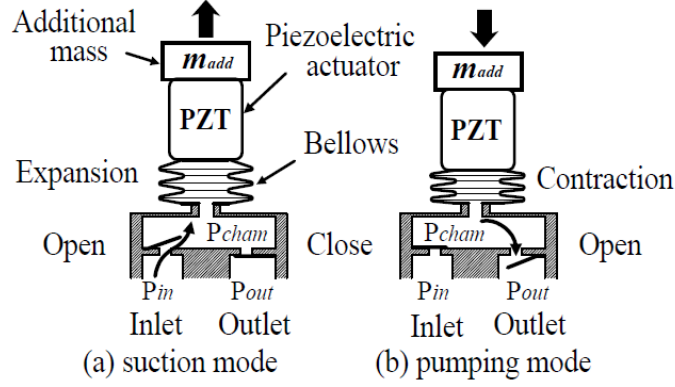


Fig. 1.9: Resonantly driven piezoelectric micropump. The produced prototype amounts 9 mm of diameter by 10 mm high, for a maximum measured flow rate of 4.8 ml min^{-1} , a maximum pressure of 320 kPa and a maximum output power of 8.7 mW, as reported in Park et al. (1999).

measured at a speed of 6000 rpm for water. Dynamics pumps are also based on principles which are not exploitable in ordinary pumps such as in electrohydrodynamic, electroosmotic and magnetohydrodynamic micropumps.

Electrohydrodynamic micropumps are based on body force appearing in dielectric fluids when an electric field E is applied. According to Laser and Santiago (2004), it is given by:

$$F_e = \underbrace{q_f E}_{\text{Coulomb force}} + \underbrace{P \cdot \nabla E}_{\text{Polarization force}} - \underbrace{\frac{1}{2} E^2 \nabla \epsilon}_{\text{Dielectric force}} + \underbrace{\nabla \left(\frac{1}{2} \rho \frac{\partial \epsilon}{\partial \rho} E^2 \right)}_{\text{Electrostrictive force}} \quad (1.1)$$

where the spatial charge density q_f , the polarization vector P , the permittivity ϵ and the mass density ρ are the ones of the fluid. As explained in il Jeong and Seyed-Yagoobi (2002), the electrostrictive force is only relevant for compressible fluid. Most of reported design for electrohydrodynamic micropumps in Laser and Santiago (2004) and Iverson and Garimella (2008) exploit the Coulomb force term. They are classed according to the effect used to produce the spatial charge density. In induction type, a gradient in electrical conductivity is required. Some possible methods to obtain it are: anisotropic heating of the fluid, discontinuities in the fluid properties by using different layers of immiscible fluids. Charges are induced at the layers' interface or in the bulk of the fluid i.e. where a gradient occurs. By applying a travelling wave, electrical charges move and carry the fluid with them by viscous effect. In conduction type, charges are produced by dissociation of molecules in the fluid, according to il Jeong and Seyed-Yagoobi (2002). Once more, the flow is due to the vis-

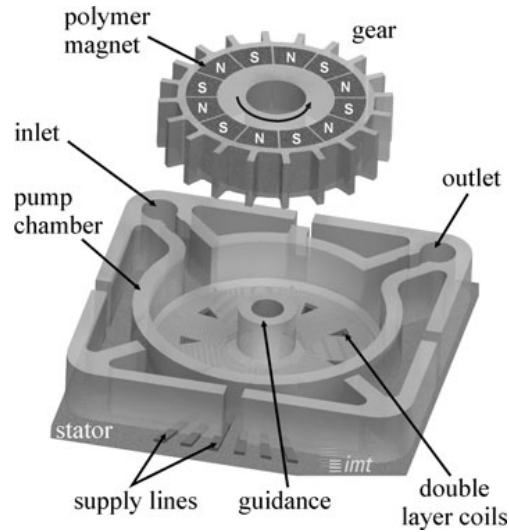


Fig. 1.10: Micro centrifugal force pump with internal synchronous micro motor. Several prototypes have been produced with rotors down to 2.6 mm in diameter and rotating speed up to 4000 rpm. Polymer magnets have been integrated in the rotor which allows a compact design. Unfortunately no measurement have been reported in Waldschik and Büttgenbach (2010a)

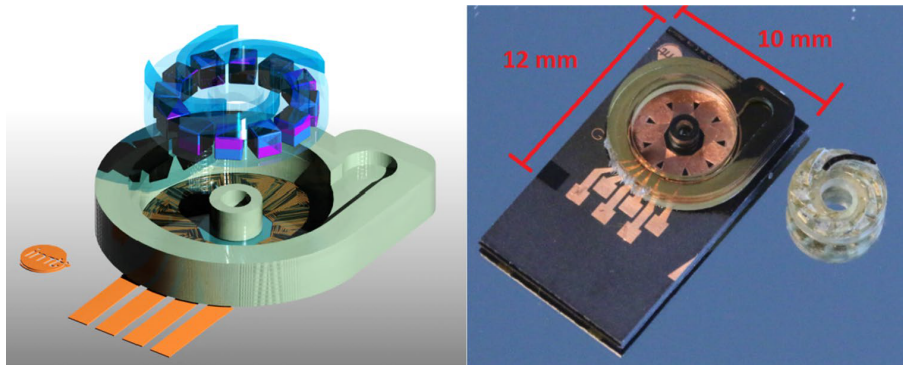


Fig. 1.11: Micro centrifugal force pump with internal synchronous micro motor. A flow rate of 14.3 ml/ min at 11 400 rpm and no back pressure, as well as a maximum pressure of 1.57 kPa at 9000 rpm have been measured. Matar et al. (2017)

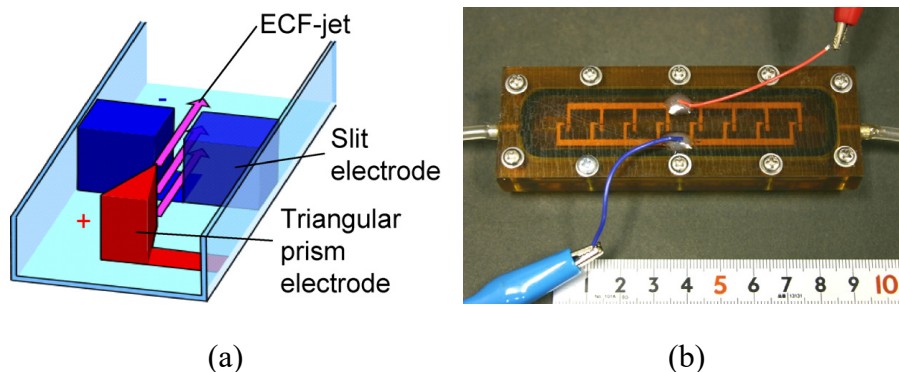


Fig. 1.12: Electro conjugate fluid jet micropump. A jet flow is created in an ECF when a high DC voltage is applied between electrodes as shown in (a). One example of it is the one in Kim et al. (2012) shown in (b) with a pressure of 53 kPa and flow rate of 53.4 ml min^{-1} .

cous effect between the charges and the fluid. In injection type, the charges are due to ions injected when a sufficient electric field is applied. However this seems to necessitate a specific couple of electrode/fluid with specific geometry to inject ions in the fluid. Once more, the fluid is transported by viscous interactions. These pumps are most of the time valveless and their directionality is given by the body force. They require a quite high electric field from 10 kV cm^{-1} to more than 100 kV cm^{-1} , for a flow rate up to 14 ml/min and a maximum pressure of 0.78 Pa , depending on the design, as reported in Laser and Santiago (2004).

Another principle based on electric field is the use of electroconjugate fluid jets. Electroconjugate fluid is a functional fluid which exhibits a jet flow when a high DC voltage is applied between electrodes. The principle is shown in 1.12(a). They also require a high voltage of about 6 kV for pressure around 53 kPa and flow rate around 53.4 ml min^{-1} for the design tested in Kim et al. (2012) and shown in fig. 1.12(b).

Electroosmotic based micropumps use the electrical double layer which appears when a liquid is put in contact with a solid. If a DC electric field is applied in the flow direction, it will interact with the surface charges and, by viscous forces, a flow is created. In the case of AC electroosmotic pumps, the electrical double layer is not due to the deprotonation of the solid surface but is rather established by the electrodes positioned on the channel boundary. According to Iverson and Garimella (2008), the most common geometry is the one with asymmetric electrodes. It induces an electric field and creates the diffusive layer charges along the electrodes' surface. These pumps' performances strongly depend on the working fluid property, such as ion density and its pH. The higher reported flow rate is 33 ml/min and for pressure, it is 20 MPa .

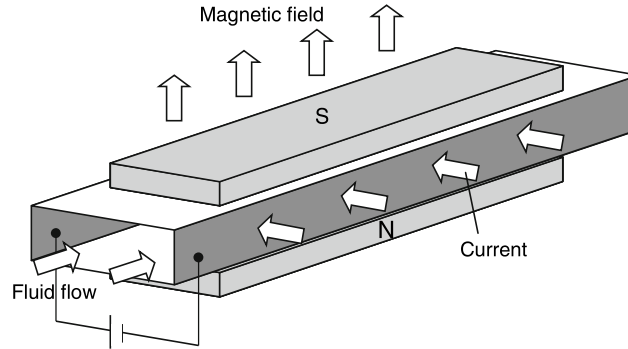


Fig. 1.13: Principle of magnetohydrodynamic micropump. A magnetic field and a current density generate a body force in a conducting fluid according to Lorentz law. This creates a flow in the direction perpendicular to the magnetic field and the current density. Figure from Iverson and Garimella (2008).

Magnetohydrodynamic pumps use a conducting fluid to carry a current that interacts with a magnetic field to displace fluid. It is in fact a fluid application of the Lorentz force, which means the fluid will move in a direction perpendicular to both current density and magnetic field. A sketch of the principle is shown in fig. 1.13. Since the fluid has to be conductive, not all fluid can be employed. In addition, Joule heating of the fluid is unavoidable, which means the application has to permit it. Some other designs, as well as explanations on various phenomena in micro-magnetohydrodynamics, are provided in Nguyen (2012). As an example of the performances that can be reached, let's take the micropump developed in Homsy et al. (2007). It uses the magnetic field of a nuclear magnetic resonance system to move fluid in a microfluidic chip. A pressure of 180 Pa and a flow rate of $1.5 \mu\text{l}/\text{min}$ are reported.

Other phenomena have been used to create micropumps, such as electrowetting which uses the modification of surface tension under the action of an electric field to generate a flow. Fluid regions with higher surface tension move to regions where it is lower. Micropumps based on synthetic jet generation, such as the one studied in He et al. (2016), have also been developed. A synthetic jet is created by a vibrating diaphragm which sucks and expels a fluid by a small orifice. This creates vortices which moves rapidly meaning they carry fluid away which will not be influenced by the next suction stage. This type of micropump is not a displacement pump since no rectification is provided meaning the mean mass flux across the orifice is null. However it transfers momentum to the fluid which is the definition of a dynamic pump. The studied design is shown in fig. 1.14.

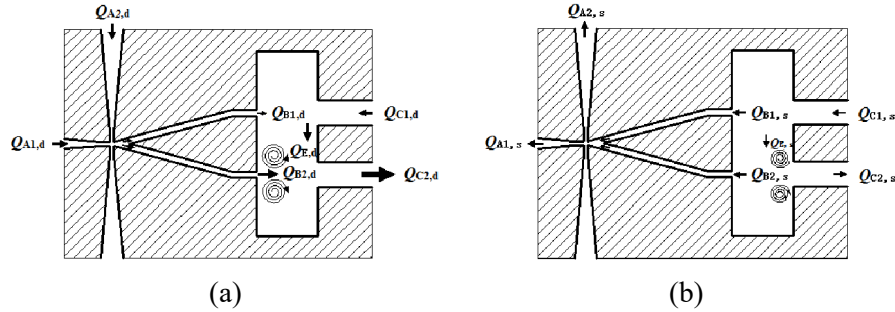


Fig. 1.14: Principle of synthetic jet micropump. A chamber is used to generate the synthetic jet, two others are used to direct it i.e. they are used as a rectification mean. The three chambers are not represented here but they are linked to the three channel on the left. Figure from He et al. (2016).

This is not an exhaustive list of all tested micropumps, but it should have given an idea of the vast choice of pumping principles and actuation mechanisms which are used in microfluidic devices. The next subsection explains several means for fluid rectification which are necessary for some of the presented pumping principles.

1.1.2 Flow rectification

Flow rectification ensures the directionality of the flow and is necessary in many designs, especially for displacement pumps. It works on the fluid flow as a diode works on electrical current. Various principles can be used to rectify the fluid. Some of them ensure a preferential path for the fluid flow and thus allow a reverse flow. Some other ones ensure fluid flows in one direction only, at least up to a certain differential pressure. So rectification does not mean a back flow is not observed but that the mean flow is not zero. Depending on the pumping principle, this action is ensured without any additional component, by geometrical design or by additional components called valves. In addition, depending on the rectification used, the pump can be used to move fluid in both directions. The pump is then bidirectional.

When no component is needed to rectify the flow, the pump is said to be valveless. In this case the rectification is somehow integrated in the working principle. Dynamic pumps are, for most of the design, part of this category. An example of displacement pump which is valveless is the diaphragm pump where several chambers are put in series and actuated in a peristaltic way, such as the one shown in fig. 1.2.

By designing proper inlet and outlet geometry, such as in nozzle diffuser or Tesla valves, the flow has a preferential path. They are shown in fig. 1.15(a) and fig. 1.15(b) respectively. This is commonly found in diaphragm pumps. It can be easily inte-

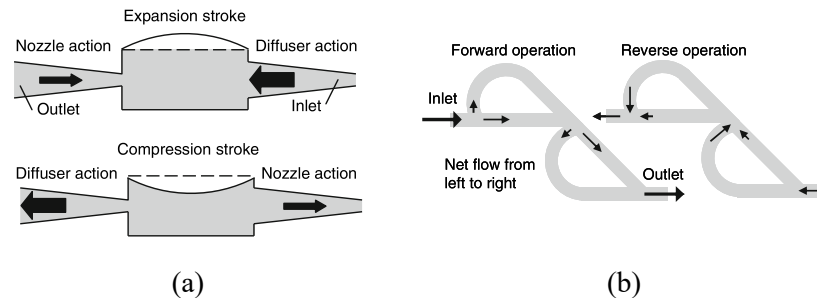


Fig. 1.15: (a) Nozzle diffuser and (b) Tesla valves. Depending if it is the suction or discharge, the flow has a different preferential path. Figure from Laser and Santiago (2004).

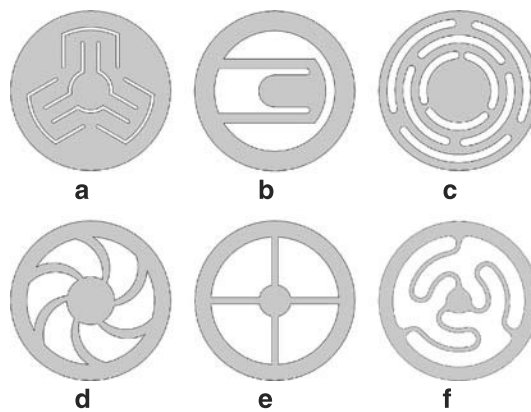


Fig. 1.16: Spider valves. Figure from Smal et al. (2008).

grated and no moving parts are involved. This has the advantage of being simple and robust. Moreover it would less likely damage the fluid, which is a desirable property in biomedical applications, such as the ones manipulating blood cells. A drawback is that a back flow is unavoidable. Moreover, these pumps are generally mono directional.

Another solution to rectify the flow is to add a one way valve which will open for one flow direction and close in the other. At least two valves must be used. They can be passive, in which case they are opened and closed by the flow itself, such as in ball valves or spider spring valves shown in fig. 1.16. They can also be active and their opening and closure is ensured by an additional actuator, such as a piezoelectric material. Additional designs and explanation can be found for example in Oh and Ahn (2006).

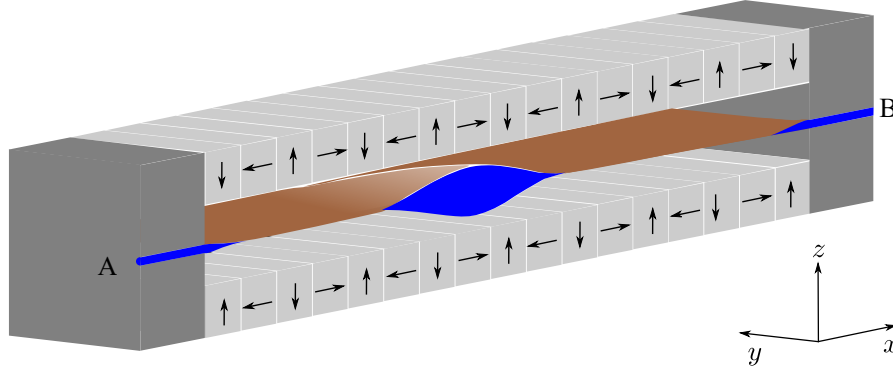


Fig. 1.17: Electromagnetic micropump, original design. The stator is composed of two arrays of permanent magnets, known as Halbach arrays, and put face to face. Fluid is transported from A to B, or inversely, between two thin diaphragms.

1.2 New micropump concepts

In this thesis, two micropumps of the displacement type are introduced. In both designs the fluid is displaced from inlet to outlet by creating a peristaltic movement of two thin diaphragms using bending stress. They are considered as micropumps because they are capable of dosing a fluid quantity under a microlitre. They are however bulkier than most of devices presented in the literature and the question of their miniaturization is not part of this thesis. These micropumps are presented more deeply in the next two subsections.

1.2.1 Electromagnetic micropump

In this concept, the peristaltic movement is created using the action of Lorentz forces to displace the diaphragm. The pump is composed of a static part, the stator, which creates a magnetic field in a thin space, hereafter called the air gap, where the two diaphragms, which are used to pump the fluid, are placed.

As shown in fig. 1.17, the stator has an upper and lower part which are arrays of permanent magnets. In each array, the magnets are placed in a special configuration, known as Halbach array, which has the particularity to strengthen the magnetic field on one side of the array by creating a preferential path for the flux. The upper and lower arrays are put face to face, thus increasing the x component of the magnetic field and at the same time reducing its z component. Along the x axis, the magnetic field is alternatively positive and negative as shown by the direction of the arrows in fig. 1.18. Hereafter, the length between two sign switches in the pole pitch is called a magnetic phase and is denominated as ψ on the figure.

The x component of the magnetic flux density, B_x , is used to move the di-

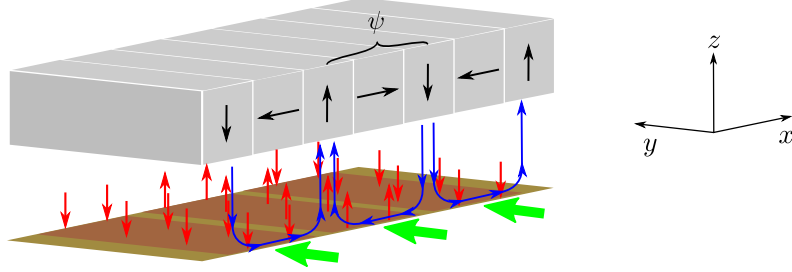


Fig. 1.18: The halfbach arrays impose a magnetic field with a strong x component and which changes of sign between two pole pitches ψ , as shown by the blue arrows. Injecting a current density in the y direction, as shown by the green arrows, in the three tracks, lead to a force density which will lift the diaphragm up for the middle track and push the two other ones as shown by the red arrows. Each track covers a pole pitch.

aphragm in z direction. These diaphragms are made of a dielectric material with electrical tracks bonded on them, which have the same width as the magnetic phase and are placed at the vertical of them. To induce the movement, a current density is imposed in these tracks, creating a force on the diaphragm in the same direction as the movement. According to Lorentz law, the current density J has to flow in the y direction to obtain the desired force density $\vec{f}$:

$$\begin{aligned}\vec{f} &= J\hat{y} \times B_x\hat{x} \\ &= -JB_x\hat{z}.\end{aligned}\tag{1.2}$$

The powering scheme is such that at every moment three phases are powered as shown in fig. 1.19 for the upper diaphragm. The current density in these phases has the same sign. Since the magnetic flux density has opposite signs between phases, the force density on the central track will move up the diaphragm to create the fluid bubble. On the two other tracks however, it will push the diaphragms against each other, ensuring the sealing of this small amount of fluid. A similar profile, but opposite in sign, deflects the lower diaphragm downward.

By powering successively the phases with a proper current pattern, such as the one proposed in table 1.1, the bubble will be transported from port A to B as shown in fig. 1.19. Conversely the powering scheme can be reversed to move the fluid from B to A. This design is thus bidirectional and valveless, since it intrinsically ensures the rectification.

The design presented here is symmetric but an asymmetric one is also possible.

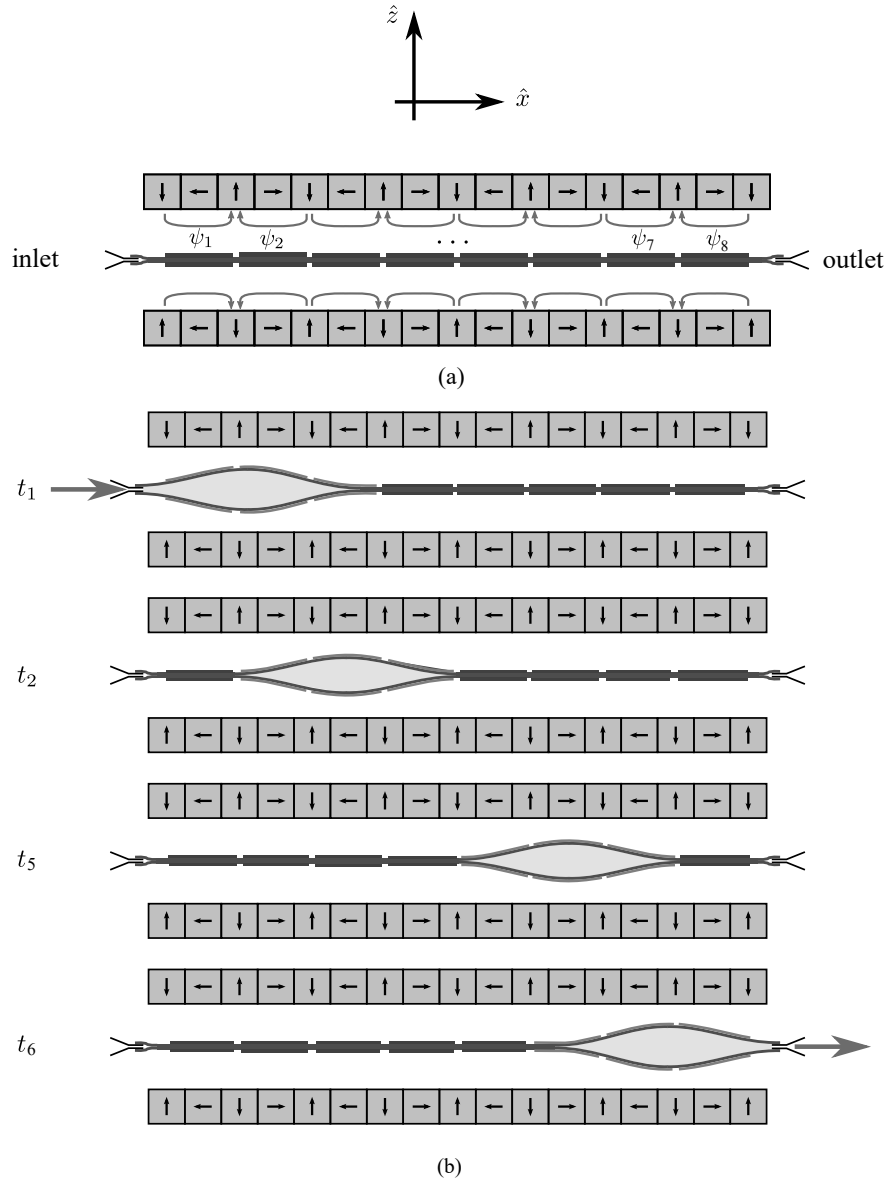


Fig. 1.19: Electromagnetic micropump working principle. A small volume of fluid is taken at the inlet and transported to the outlet by creating a peristaltic motion of the diaphragm. This is done by bending the diaphragms under the action of Lorentz forces. Powered electrical tracks are the one in yellow.

	Ψ_1	Ψ_2	Ψ_3	Ψ_4	Ψ_5	Ψ_6	Ψ_7	Ψ_8
t_1	J	J	J	0	0	0	0	0
t_2	0	$-J$	$-J$	$-J$	0	0	0	0
t_3	0	0	J	J	J	0	0	0
t_4	0	0	0	$-J$	$-J$	$-J$	0	0
t_5	0	0	0	0	J	J	J	0
t_6	0	0	0	0	0	$-J$	$-J$	$-J$

Table 1.1: A typical powering scheme of the electromagnetic micropump upper diaphragm of fig. 1.19. J is the amplitude of the current density, a positive value flowing in the $\hat{y}$ direction.

In this case, the stator is made of one array of magnets and one diaphragm. This is like cutting in half the pump shown in fig. 1.17 at the xy symmetry plane and replacing the other part by a flat part made of a non-ferromagnetic material. Loosing the symmetry has however a cost. The volume will be divided by two since only one diaphragm is used but in addition the magnetic field will be weaker and thus the diaphragm deflection will be smaller. Using two diaphragms also helps to reduce mechanical wear because at the contacting zone and in case of dry contact, the relative motion between both diaphragms is zero in a perfect symmetric case. One of the advantage could be that the asymmetric pump is less bulky. Indeed, the repulsive force due to two arrays in the symmetric design require a thick housing to reduce its bending. In addition introducing the fluid should be easier in the asymmetric case since it can be done by drilling a hole in the lower part of the housing. In the symmetric case, fluid has to be injected between the two diaphragm without loosing the contact of the diaphragms at rest state and with ensuring static sealing at the injection and ejection port which is more complicated.

1.2.2 Piezoelectric micropump

The piezoelectric micropump has a similar working principle but actuation is rather different. It uses a layer of piezoelectric material, which deforms under the action of an electric field, to bend the diaphragm on which it is bonded, as shown in fig. 1.20. By applying a space and time varying electric field along the main axis x , a peristaltic movement is created which move the fluid from inlet to outlet.

The electric field is due to the application of a voltage difference on the electrodes fixed on top and bottom of the piezoelectric layer. In order to create an electric field which varies in space, the upper electrode is segmented in several phases ϕ . The piezoelectric layer is thus made of one piece of material, polarized in the out of plane direction of the layer i.e. the thickness direction, z . The application of an electric field along the same axis stretches or expands the piezoelectric layer and, since it is bonded to diaphragm, creates a bending movement of it.

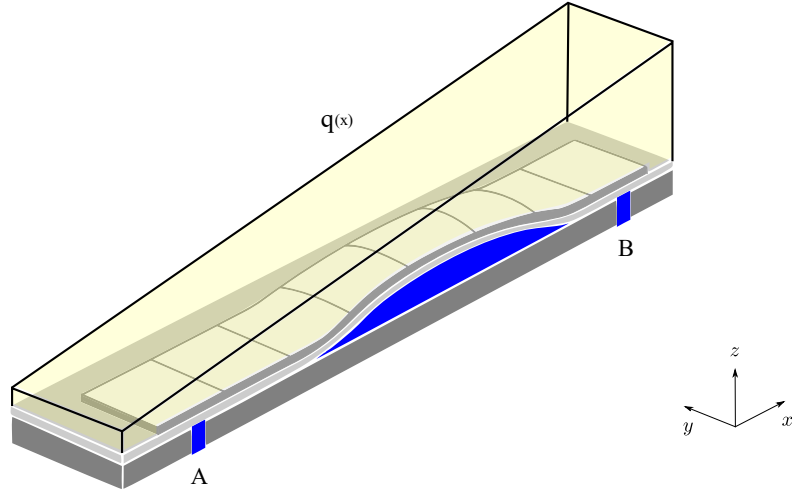


Fig. 1.20: Design of the piezoelectric micropump with one diaphragm, original design. Fluid is transported from A to B, or inversely, between the thin diaphragm and the bottom of the housing of the micropump housing. Sealing is ensured by an applied external pressure $q(x)$ which is represented here in light yellow.

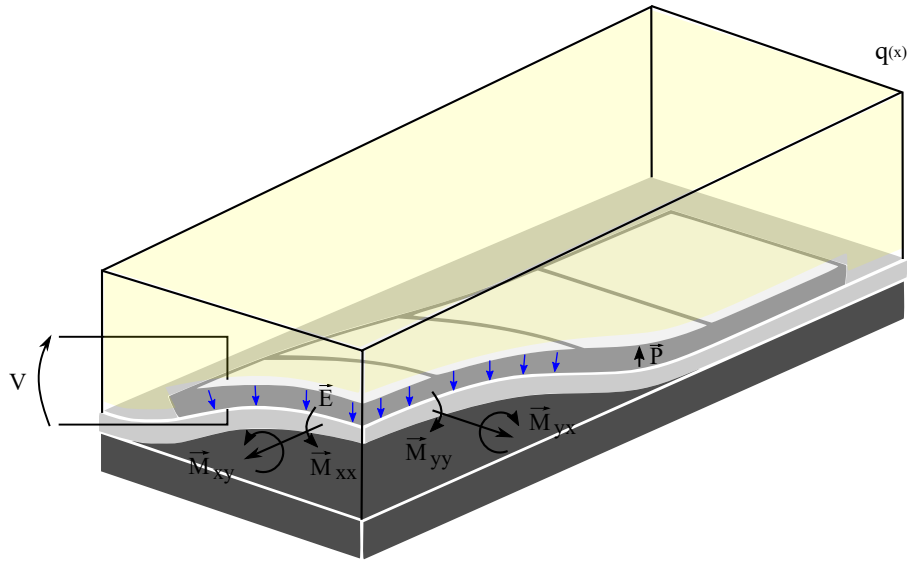


Fig. 1.21: Loads applied to the diaphragm. The moments M_{ij} are the result of the applied external pressure $q(x)$, represented in light yellow, and of the piezoelectric effect due to the d_{31} coupling coefficient. An applied electric field $\vec{E}$, represented by blue arrows, in the opposite direction compared to the polarization $\vec{P}$, expands the piezoelectric layer and bends the diaphragm.

	ϕ_1	ϕ_2	ϕ_3	ϕ_4	ϕ_5	ϕ_6	ϕ_7	ϕ_8
t_1	V	V	V	0	0	0	0	0
t_2	0	V	V	V	0	0	0	0
t_3	0	0	V	V	V	0	0	0
t_4	0	0	0	V	V	V	0	0
t_5	0	0	0	0	V	V	V	0
t_6	0	0	0	0	0	V	V	V

Table 1.2: Powering scheme of the piezoelectric micropump with the applied voltage V .

As in the electromagnetic pump, three phases are powered at the same time following the powering scheme given in table 1.2, creating a lifting of the diaphragm as shown in fig. 1.22. However, in the electromagnetic pump, the powering also ensure the sealing of the fluid bubble by pushing at the extremities of the bubble and forcing the contact between the diaphragms. In the piezoelectric micropump, the diaphragm is pushed against the bottom of the pump by applying an external pressure on top of the diaphragm. In order to increase the mean pressure in the fluid when the bubble is moved in the pump and thus that each section of the piezoelectric patch works at constant mechanical work, the external pressure increase linearly from inlet to outlet. Fig. 1.21 shows all the loads applied to the diaphragm.

The presented design, in this case, is the asymmetric one, using only one diaphragm. As for the electromagnetic pump, the bottom of the pump can be replaced by another diaphragm, actuated in the same fashion. It will theoretically double the volume of the pumped bubble and reduce the mechanical wear thanks to the fact that there is no relative displacement between the diaphragms, at their contacting surfaces.

1.2.3 Comparison of concepts

The two designs presented in the last section are original concepts of normally closed micropumps, which use travelling bending movement of diaphragms to move fluid from one port to the other. The term ‘normally closed’ means that, at rest, the diaphragm is in contact with another diaphragm or with the housing of the micropump, depending if the design is symmetric or asymmetric. The reverse flow is avoided by ensuring the contact between the diaphragm and the bottom of the micropump, in normal operation, such that inlet and outlet port are never linked by a fluid channel. It means the micropumps need no valve to work properly and it simplifies the mechanical design. The drawback is that the contact could damage some fluids, such as blood, discarding these designs for such fluid handling.

Micropumps only differ, in their principle, by the actuation and by the way the contact is enforced. In the electromagnetic micropump, actuation and contact is

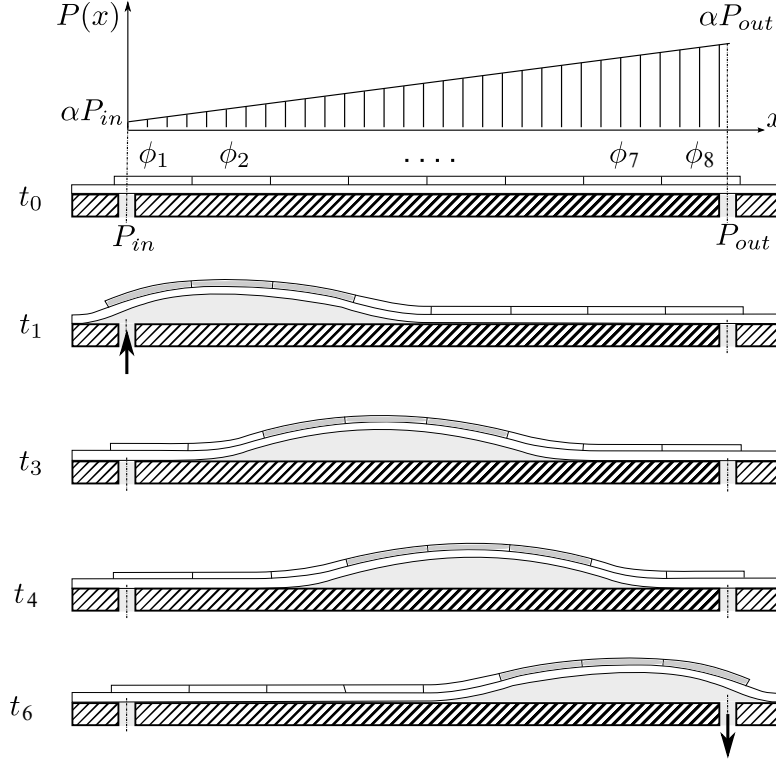


Fig. 1.22: Design and working principle of the piezoelectric micropump with one diaphragm. Phases ϕ in grey are the ones which are powered.

due to the electrical current which is imposed in the tracks, which interact with the surrounding magnetic field. In the piezoelectric micropump, the actuation is due to the electric potential and piezoelectric effect in some layers of the diaphragm. Contact is enforced by an external force density whose implementation will not be discussed here.

From a modelling perspective, the similarities of both designs mean it should be possible to reproduce their behaviour through a unified framework, using the same tool. It has to be able to represent faithfully the kinematics, or dynamics, of the diaphragm, including the contact with another diaphragm or with the housing of the micropump but also the dynamics of the fluid which is transported.

1.3 Thesis objectives

Performing simulations helps to understand how a system works. They can be used to investigate the performances of the system but also to evaluate things which can not be measured directly such as the maximum value of the stresses or electrical field in the material. For this purpose, a theory which takes into account the main physical phenomena is essential. The model can be used for optimizing the devices, which means that it will be evaluated many times, requiring an efficient tool. Here, efficient means choosing the right balance between the phenomena which are modelled and the computational cost and memory requirements. It should give a good idea of the displacement of the diaphragm. In addition, an accurate knowledge of the stresses and electric field in the material is required, especially in the piezoelectric material, where exceeding these values will destroy the device.

So the main objective is to create efficient tools to model devices, such as the proposed micropumps which will be used as benchmark cases. The main characteristic of these devices is that they involve a thin moving part which can be composed of several layers of different materials. It can come into contact with a rigid obstacle. A thin layer of fluid is also transported in the micropumps, but here, the fluid and its coupling to the structure will not be considered.

1.4 Research contributions

In order to reach the presented objectives, several research questions have appeared and are answered.

The modelling structures with at least one dimension smaller than the other.

- One dimensional models based on laminated beam theory are developed. They are firstly compared to a simple test case, the piezoelectric bending actuator. Then they are applied to the micropump case, showing its ineffectiveness in modelling faithfully the stresses, but also the displacement in some situations.
- Two dimensional models for laminated plate dynamics, based on Kirchhoff-Love assumptions, are developed. This theory compares well to the three-dimensional finite element models for the piezoelectric bending actuator but also for the micropump benchmark models.
- Evolution of the electric field across the thickness in piezoelectric layer and its impact on the stresses are determined. An expression for it is developed, consistent with the electrostatic conservation laws and the equipotential condition imposed by the electrodes on the piezoelectric layers.

- Piezoelectric effect and Lorentz force density are integrated in the two theories through equivalent bending moments and surface force density, respectively.

This has led to an efficient model where coupling between bending and stretching is taken into account. In addition, the electrical equations do not have to be solved since an exact expression for the electric field is available, satisfying the electrostatic laws. This gives a theory where displacement components are the only unknowns.

The contact constraint formulation for beam and plates equations is studied and an algorithm based on active set strategy is given to tackle the problem.

An algorithm which solve the laminated plates dynamics, for the displacement degrees of freedom, taking into account frictionless contact constraint, piezoelectric effect, Lorentz force and mechanical loads, is developed based on the finite element method. The electric field in piezoelectric layers and stresses are computed in a post processing step.

1.5 Thesis outline

The text is divided in two parts. The first one is dedicated to the one dimensional models, where the structure is studied as a beam actuated such that quasi-static assumptions are verified. Constitutive equations for composite beams made of several layers where some are in piezoelectric material are derived in chapter 2. They are validated on a simple case study before being applied to the micropumps in chapter 3. These drastic assumptions lead to simple equations which are tractable by analytical methods. The results are then analysed and compared to models developed in 3D finite element model, using a commercial software, to show the limits of the one dimensional theory.

The second part develops the equations for two dimensional models. The chapter 4 develops the laminated plate theory where some laminates are made of piezoelectric layers or subjected to Lorentz force. A numerical scheme based on finite element method is described and used to analyse the dynamics of a bender. Comparison to a 3D model based on a commercial software shows the effectiveness of the scheme. In addition, a method for dealing with contact constraint based on an active set strategy is presented in chapter 5 where a comparison to a 3D model is also made. These tools are applied to the two micropump benchmark cases in chapter 6.

Part I

Modelling - One dimensional approach

Constitutive equations for laminated Euler-Bernoulli beams

2

In this chapter, the theory for laminated, long, thin and narrow structures, bent under external pressure or piezoelectric effect, in quasi-static conditions, is introduced. It is based on Euler-Bernoulli assumptions for beams and will be called laminated beam theory. The aim is to obtain simplified equations, which renders faithfully the underlying physics, to accurately evaluate the displacement, stresses and electric field. Compared to theories already introduced in the literature and overviewed in the first section, the one developed here integrates the effect of the variation of the electric field across the thickness of the piezoelectric layers on the stresses, which, as will be shown, has a noticeable impact. However, this variation is supposed to have a second order impact on the deflection and is thus neglected for its computation. Then, the developed equations are used to analyse the bending of a piezoelectric bender made of an elastic layer and a piezoelectric one. The results are compared to a 2D model developed in Comsol Multiphysics, which will be described. The results presented here have been the subject of Beckers and Dehez (2013a), *Modelling of electric field and stresses in piezoelectric composite under bending load in quasi-static conditions*.

2.1 Overview of literature

Looking at the literature, it appears that the modelling approach is strongly dependent of the purpose. People dealing with the design of devices, such as piezoelectric benders or Lorentz force based actuators or sensors and their optimization, usually seek very simple models which render reasonably the underlying physics. At the opposite, there are people who seek to reproduce faithfully the behaviour of the laminated structure, which leads to more involved models to implement.

They differ by the assumptions used to develop the theory, by the effects which are taken into account and by the proposed method used to solve the equations. The most recent articles usually treat dynamical problems, take into account nonlinear effect using von Kármán strains and are solved using finite element method, such as in Lazarus et al. (2012), Jabbari et al. (2016b) or Jabbari et al. (2016a).

Some authors have also published articles where 3D effects are introduced through modified beam coefficient such as in Maurini et al. (2006b), Maurini et al. (2006a) or Fernandes and Pouget (2010). In these articles, they tried to replace classical assumptions used to take into account effect along the width coordinate, i.e. null yy -stress or null y -strain, and compare them to 3D model based on finite element method.

In Krommer (2001), the full electromechanical coupling, for various electrical boundary conditions, is taken into account through effective stiffness coefficients and applied to laminated piezoelectric beam under Euler-Bernoulli beam assumptions. The electric field variation along the thickness coordinate is thus taken into account. This coupling and its link to the displacement expression has also been studied for more advanced beam theory such as in Gopinathan et al. (2000).

When it comes to articles dealing with modelling of devices, they are either based on finite element software or on simple theories which allow to treat the problem analytically such as in Nadal et al. (2017). Some articles using beam theory to model benders, monomorphs or bimorphs structures have also been published. One of the first paper published on the subject was Smits et al. (1991), which used an energy approach for very specific cases. More recently Dunsch and Breguet (2007) used a superposition method to take into account mechanical loading and electrical loading separately. Electric field was supposed constant across thickness. This assumption is inconsistent with respect to the electrical equations but it gives a fairly good result to evaluate displacement. However, in applications, designers are also interested in the stresses, especially for piezoelectric ceramics which are brittle, and electric field value to avoid depolarization. In addition, the impact of the electric field on the stresses value is significant. The approach has thus been improved in Dehez (2011) where an expression is given for the electric field along the thickness coordinate and its impact on the stresses is considered. In this part, the same approach is used and extended to multilayer structure with piezoelectric layer and subjected to electromagnetic loads.

2.2 Bending of composite beams

As shown in fig. 2.1, the structure under consideration is composed of N layers of different materials, with constant thickness. It is assumed that the bonding between each of them is ideal, which means it ensures strains continuity. The structure is supposed to have two characteristics length, its width, along y axis, and thickness, along z axis, much more smaller than the third one, called length, along the x axis. In such

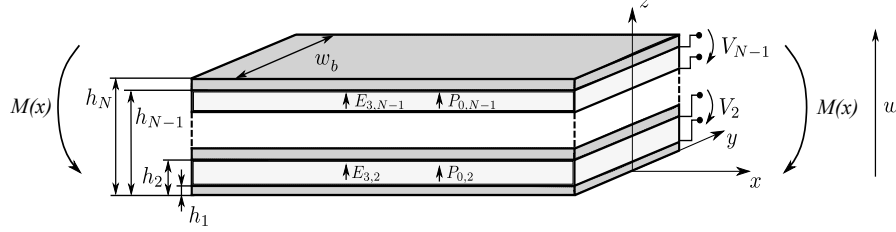


Fig. 2.1: Laminated structure under mechanical and electrical loads. h_k is the cumulative thickness of the k first layers, $M(x)$, the internal bending moment, w , the deflection and w_b , the width of the beam. Piezoelectric layers are submitted to a voltage V and are polarized in the z direction.

a structure, a virtual line, usually called neutral fibre, which does not experience any strain, is defined by its position, z_0 , along the z axis. In this theory, it is supposed that the deformations of the structure remain small and that Euler-Bernoulli assumptions are valid, which means straight lines, normal to the neutral fibre, are supposed to remain straight and normal to it after deformation. This allows to express the deformation, or strain, in x direction, S_{xx} as:

$$S_{xx} = \frac{1}{\kappa} (z - z_0) \quad (2.1)$$

where κ is the curvature radius, generally dependant of the x coordinate. These strains are due to two different effects here, which are the internal bending moment, due to external force and pressure loads, and piezoelectric effect, due to an applied voltage. Only linear constitutive equations of materials will be considered, which means that, since the strain is also linear, each effect can be considered separately. They are characterised by a curvature radius, κ_m and κ_ν , and a neutral fibre position, $z_{0,m}$ and $z_{0,\nu}$, for external force or pressure and piezoelectric effect, respectively. The deflection of the beam, $w(x)$, is obtained by integrating twice the total curvature:

$$w(x) = \int_0^x \int_0^{x'} \frac{1}{\kappa_m} + \frac{1}{\kappa_\nu} dx'' dx'. \quad (2.2)$$

In order to obtain an expression for this curvature and the position of the neutral fibre, the static equilibrium for the beam is considered and conditions on forces and moments are summarized as:

$$\sum \vec{F} = 0 \quad (2.3)$$

$$\sum \vec{M} = 0 \quad (2.4)$$

where $\vec{F}$ are the forces and $\vec{M}$, the bending moments. These equations must be verified globally and locally. In addition, internal forces $F(x)$ and moments $M(x)$

are linked to the strain, through the stress, by:

$$F(x) = w_b \sum_{i=1}^N \int_{h_{i-1}}^{h_i} T_{xx} \, dz \quad (2.5)$$

$$M(x) = w_b \sum_{i=1}^N \int_{h_{i-1}}^{h_i} z T_{xx} \, dz \quad (2.6)$$

where w_b is the width of the beam and T_{xx} , the local stress with respect to the x axis. It is linked to the strain by the constitutive law of the material used. Considering a thin narrow beam, the only interesting stress component will be the one in x direction, all other components can be neglected. For piezoelectric material, since they are used in bending modes, the polarization is in the thickness direction and thus, only the coupling between the z and the x axis is used. These relations can be written as:

$$S_{xx} = s_{11} T_{xx} \quad (2.7)$$

$$S_{xx} = s_{11} T_{xx} + d_{31} E_z \quad (2.8)$$

where s_{11} is known as compliance coefficient and d_{31} , as piezoelectric coefficient. E_z is the electric field along z axis. The first equation is for linear elastic material and the second for linear piezoelectric material. These equations can now be combined to find the curvature and neutral fibre position. As was said earlier, each effect is considered separately so, let us start with the external force density and punctual force only. As will be shown latter, the curvature will induce an electric field in the piezoelectric material, which varies linearly with respect to the thickness. It is assumed that it has only a second order effect on the deflection and is thus neglected for its computation. Only the mean value of the electric field is kept and is here null. The beam is supposed to be loaded with external force density and punctual force applied along the z axis, but with no load along the x axis. Thus, $F(x)$ in (2.5) is null and the loads will be compensated by an internal moment $M(x)$ of (2.6). Using these two equations with (2.7) and (2.8) leads to:

$$0 = \frac{1}{\kappa_m} (A_2 - A_1 z_{0,m}) \quad (2.9)$$

$$M(x) = \frac{1}{\kappa_m} (A_3 - A_2 z_{0,m}), \quad (2.10)$$

where A_k are defined for $k = 1, 2, 3$ as:

$$A_k = \sum_{i=1}^N \frac{1}{s_{11,i}} \left(\frac{h_i^k}{k} - \frac{h_{i-1}^k}{k} \right). \quad (2.11)$$

Combining (2.9) and (2.10), an expression for the curvature and neutral fibre position is found:

$$\frac{1}{\kappa_m} = \frac{-M(x)A_1}{w_b(A_2^2 - A_3A_1)} \quad (2.12)$$

$$z_0 = \frac{A_2}{A_1} \quad (2.13)$$

The curvature is thus dependent of the internal moment which depends of the applied loads and of the boundary conditions, which both depends on the problem. Considering now the piezoelectric effect, all applied pressure or forces are put to zero, which means that both $F(x)$ and $M(x)$ are null. In this case of course, the electric field E_z is not null and is put to its mean value, which for the i th layer is given by:

$$E_{z,i} = -\frac{V_i - V_{i-1}}{h_i - h_{i-1}}. \quad (2.14)$$

Applying the same reasoning as for the mechanical loads, using (2.5), (2.6), (2.7) and (2.8), two equations are obtained:

$$0 = \frac{1}{\kappa_\nu} (A_2 - A_1 z_{0,\nu}) + \sum_{i=1}^N d_{31,i} (V_i - V_{i-1}) \quad (2.15)$$

$$0 = \frac{1}{\kappa_\nu} (A_3 - A_2 z_{0,\nu}) + \sum_{i=1}^N d_{31,i} \frac{V_i - V_{i-1}}{2} (h_i + h_{i-1}). \quad (2.16)$$

Combining these two equations, an expression for the curvature and neutral fibre position, for piezoelectric load, is found:

$$\frac{1}{\kappa_\nu} = \frac{A_2 \sum_{i=1}^N \frac{d_{31,i}}{s_{11,i}} (V_i - V_{i-1}) - A_1 \sum_{i=1}^N \frac{d_{31,i}}{s_{11,i}} \frac{V_i - V_{i-1}}{2} (h_i + h_{i-1})}{A_2^2 - A_3 A_1} \quad (2.17)$$

$$z_0 = \frac{\sum_{i=1}^N \frac{d_{31,i}}{s_{11,i}} V_i \left[\frac{A_2}{2} (h_i + h_{i-1}) - A_3 \right]}{\sum_{i=1}^N \frac{d_{31,i}}{s_{11,i}} V_i \left[\frac{A_1}{2} (h_i + h_{i-1}) - A_2 \right]}. \quad (2.18)$$

2.2.1 Null yy -strain assumption

All these equations have been derived considering a narrow beam which can expand freely along y axis and thus T_{yy} is null, here after called null yy -stress assumption. At the other extreme, a large beam can be considered as restrained along this direction and thus it is S_{yy} which is now null, here after called null yy -strain assumption. Considering this case, the stress-strain relation now becomes:

$$S_{xx,i} = s_{11,i} T_{xx,i} + s_{12,i} T_{yy,i} \quad (2.19)$$

$$0 = s_{21,i} T_{xx,i} + s_{22,i} T_{yy,i} \quad (2.20)$$

for elastic material and:

$$S_{xx,i} = s_{11,i} T_{xx,i} + s_{12,i} T_{yy,i} + d_{31,i} E_z \quad (2.21)$$

$$0 = s_{21,i} T_{xx,i} + s_{22,i} T_{yy,i} + d_{32,i} E_z \quad (2.22)$$

for the piezoelectric one. Since these materials are at least transversely isotropic, with respect to the z axis, $s_{11,i} = s_{22,i}$, $s_{12,i} = s_{21,i} = -\nu_i s_{11,i}$ and $d_{31,i} = d_{32,i}$. From (2.20) and (2.22), an expression for $T_{yy,i}$ is obtained and replaced in (2.19) and (2.21), respectively, giving:

$$S_{xx,i} = s_{11,i} (1 - \nu_i^2) T_{xx,i} \quad (2.23)$$

$$S_{xx,i} = s_{11,i} (1 - \nu_i^2) T_{xx,i} + d_{31,i} (1 + \nu_i) E_z. \quad (2.24)$$

This means that the previously derived expression for curvatures and neutral fibre position remains the same, provided that the compliance $s_{11,i}$ and piezoelectric coefficient $d_{31,i}$ are replaced by effective ones given by:

$$s_{11,i}^* = (1 - \nu_{12,i}^2) s_{11,i} \quad (2.25)$$

$$d_{31,i}^* = (1 + \nu_{12,i}) d_{31,i} \quad (2.26)$$

2.3 Electric field study

As previously said, the deflection due to the piezoelectric effect is supposed to be due to the mean electric field. In literature, the consideration usually stops there and the electric field is assumed constant over the layer. In fact this is not in agreement with the electrical equilibrium where:

$$\nabla \cdot \vec{D} = 0 \quad (2.27)$$

should be verified. Neglecting side effects and since the layers are quite thin, this simplifies as:

$$\frac{\partial D_z}{\partial z} = 0. \quad (2.28)$$

Considering the expression given by the constitutive equation of the piezoelectric material for D_z and reordering the terms leads to:

$$\frac{\partial E_{z,i}}{\partial z} = \frac{d_{31,i}}{d_{31,i}^2 - \epsilon_{33,i}^T s_{11,i}} \left(\frac{1}{\kappa_\nu} + \frac{1}{\kappa_m} \right) \quad (2.29)$$

for null yy -stress. For null yy -strain it is:

$$\frac{\partial E_{z,i}}{\partial z} = \frac{d_{31,i}^*}{d_{31,i}^{*2} + d_{31,i}^2 (1 - \nu_{12,i}^2) - \epsilon_{33,i}^T s_{11,i}^*} \left(\frac{1}{\kappa_\nu} + \frac{1}{\kappa_m} \right) \quad (2.30)$$

In both cases the electric field is shown to vary linearly across the thickness of the piezoelectric material, since (2.29) and (2.30) are independent of the z coordinate. To obtain an expression for the electric field one has to remind that it is linked to the electric potential ϕ by:

$$E = -\nabla \phi. \quad (2.31)$$

Since only the third component is non-zero, it gives for the i^{th} layer:

$$E_{z,i} = \frac{\partial E_{z,i}}{\partial z} \left(z - \frac{h_i + h_{i-1}}{2} \right) - \frac{V_i - V_{i-1}}{h_i - h_{i-1}}. \quad (2.32)$$

Even if the electric field varies linearly across the thickness it is supposed to have a second order effect on the deflection. As will be shown later, its variation across the thickness can be quite large and its impact on the stresses can not be neglected. This means that if a constant, mean electric field is used for the computation of the deflection, the stresses are computed using (2.32) for the electric field. The constitutive equations of the beams are now fully known.

2.4 Lorentz loads

The developed theory can be employed to study beams which bends under Lorentz force density. Those forces arise from the conjunction of a magnetic field and a current density:

$$\vec{f} = \vec{J} \times \vec{B}. \quad (2.33)$$

This force density is integrated over the thickness of the beam, leading to a pressure load which is applied as for any other externally applied pressure load. If the magnetic field is not constant, the pressure will be dependant on the deflection of the beam, leading most of the time to a nonlinear problem. This evolution of the magnetic field with respect to the beam deflection is not considered in the modelling of the electromagnetic micropump benchmark case and thus, the problem remains linear.

2.5 Case study - the piezoelectric bender

In order to verify the equations developed for composite beams, and especially the integration of the piezoelectric effect, they are applied to the case of a monomorph clamped at one end and subjected to a punctual force at the other. This model has been compared to a 2D finite element model developed in Comsol Multiphysics software to check the analytical expressions of the stresses and electric field but also to show the importance of taking the evolution of the electric field across the thickness and its impact on the stresses in the design of bending actuators and sensors.

2.5.1 2D model

The bender is 20 mm long and composed of two layers of 0.5 mm, one made of AL 2014-T6 aluminium which is modelled by linear elasticity and a piezoelectric one made of PZT-5H, modelled by linear piezoelectricity. The characteristics of those materials are given in appendix A. Displacement is fully constrained at one point, numbered 1 on the figure, and only along the x axis at point 2. A constant force is imposed at point 3. A constant voltage is imposed at edge b and electrically grounded at edge a. Two conditions can be imposed to the displacement in the y direction with Comsol: the plane stress or plane strain, which correspond respectively to the null yy -stress or null yy -strain assumptions developed earlier. A triangular mesh is used and the default solver proposed by Comsol is used for a stationary study.

2.5.2 Numerical results

Fig. 2.3 shows the electric field value across the thickness in the middle of the length of the bender. Compared to the mean value of the electric field, the maximum value is up to 40 percent higher in the plane strain assumption. In addition, the stress T_{xx} has been computed using a constant mean value for the electric field and compared with the stress computed as described earlier, considering the evolution of the electric

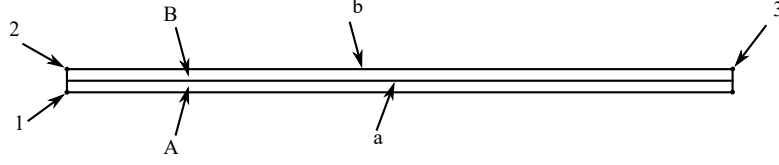


Fig. 2.2: Geometry for Comsol model. Domain B is modelled by linear piezoelectricity, with PZT-5H piezoelectric ceramic, and domain A by linear elasticity, with Al 2014-T6 aluminum. Edge a is electrically grounded and edge b is put to a fixed voltage of 50 V. Displacement at point 1 is fully constraint. At point 2, only the displacement component in x is constrained. A fixed load of 0.2 N is imposed at point 3.

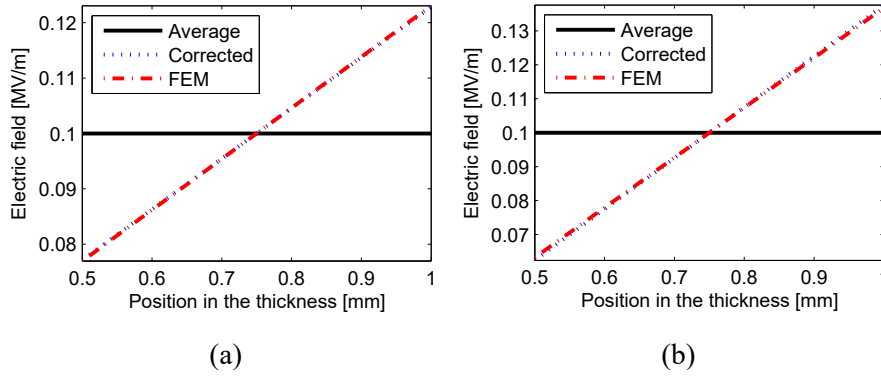


Fig. 2.3: Evolution of electric field in thickness direction in (a) null yy -stress or (b) null yy -strain assumptions at middle length. Average means here that the electric field is assumed constant across the thickness. Corrected is the one computed with the composite beam equations. FEM is the result of Comsol Multiphysics software.

field across the thickness. As shown in fig. 2.4, the stress computed with the electric field, which varies linearly across the thickness, is in better agreement with the finite element method. If the difference is not too large in the plane stress assumption, it is higher in the plane strain hypothesis. The effect is much more important on the T_{yy} stress in the plane strain assumption, as shown in fig. 2.5, where a difference of 100 percent is noticed.

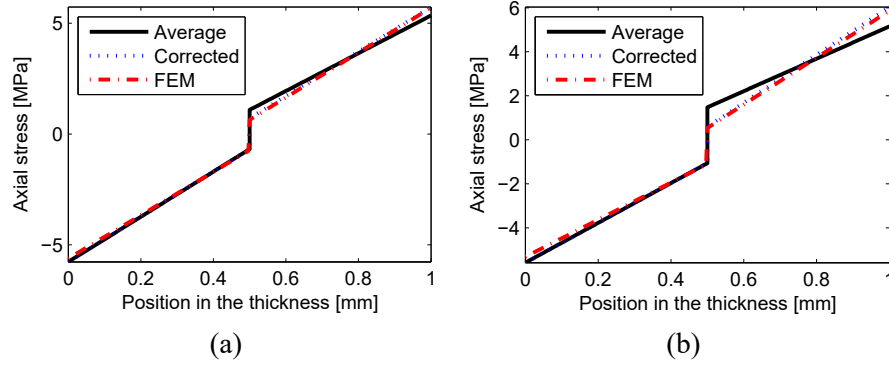


Fig. 2.4: Evolution of T_{xx} stress component in thickness direction in (a) null yy -stress or (b) null yy -strain assumptions at middle length. Average means here that the stresses are computed assuming a constant electric field across the thickness. Corrected is the one computed with the composite beam equations. FEM is the result of Comsol Multiphysics software.

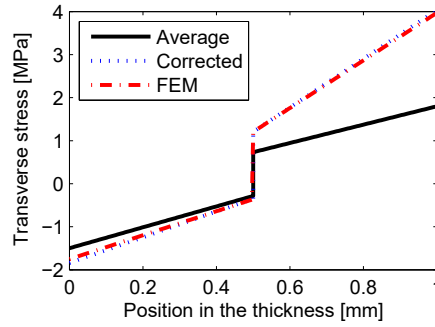


Fig. 2.5: Evolution of T_{yy} stress component in thickness direction in null yy -strain assumption at middle length. Average means here that the stresses are computed assuming a constant electric field across the thickness. Corrected is the one computed with the composite beam equations. FEM is the result of Comsol Multiphysics software.

2.6 Conclusion

In this chapter, constitutive equations for long thin structures composed of several layers of different materials are studied using Euler-Bernoulli beam theory and superposition principle as in Dunsch and Breguet (2007). It takes into consideration the bending due to an external pressure or a piezoelectric effect based on d_{31} coupling coefficient in quasi-static conditions. It follows the same idea as in Dehez (2011) for the monomorphs and considers a constant electric field on the piezoelectric patch volume for displacement computation. In a second step, an expression for the electric field, compliant with electrical equilibrium, is developed and used to compute the stresses. Those articles dealt only with two layers structures whereas here coefficients for multilayer structure have been derived. In addition, the bending under Lorentz force has also been introduced. Equations have been used to compute the stresses and the electric field in the layers in the case of a simple monomorph structure. It has been compared to the case where the evolution of the electric field across the thickness is not considered.

The results were in good agreements with the 2D finite element model developed in Comsol Multiphysics software, for stresses and electric field and the deflection will be studied in the next chapter. They have highlighted the need of taking into account the electric field evolution across the thickness and its impact on stresses in the piezoelectric layers when designing bending structures with piezoelectric materials. Indeed piezoelectric materials are brittle. A good approximation of stresses is thus essential for a proper sizing of structures involving such materials. In addition overcoming the coercive electrical field leads to depolarization of the material which has to be avoided. A good approximation of the electrical field is thus necessary too.

Application to micropump concepts

3

In this chapter, equations previously developed for composite beams are applied to the micropump concepts. The fluid is assumed to impose only a constant pressure on the diaphragm and dynamics are completely neglected, for both fluid and beams. The chapter is divided in two sections, one for each micropump. Dedicated analytical models for each micropump are obtained and explained. The 2D finite element models used for comparison are also described. This will show the good agreement between the 1D theory developed in the previous chapter and 2D finite element model which do not use specific assumptions for the different field expression across the layer thickness. The chapter finishes by showing the ineffectiveness of both analytical and 2D finite element to model the micropumps. This is based on the 3D simulation performed with a finite element method for the piezoelectric micropump. Most of results presented here have been published previously in Beckers and Dehez (2013a) and Beckers and Dehez (2014a).

3.1 Electromagnetic micropump

As previously explained, the electromagnetic pump uses the Lorentz force to bend the diaphragms and to ensure the sealing. The transverse force is due to the interaction between the electrical current in the diaphragm and the surrounding magnetic field. The volume force density is integrated over the thickness to obtain a surface force density. Thus, the modelling involves two physics: electromagnetism and structural mechanics, constrained by mechanical contact. In addition, the fluid is supposed to exert a known constant pressure on the diaphragm, which is modelled using theory of composite beams. Considering the symmetries, only a part of one of the diaphragms has to be studied, if proper boundary conditions are used. So, the considered geometry for the model is the one shown in fig. 3.1.

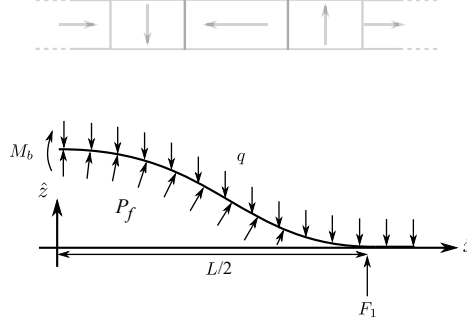


Fig. 3.1: Geometry of the 1D model taking into consideration the symmetries.

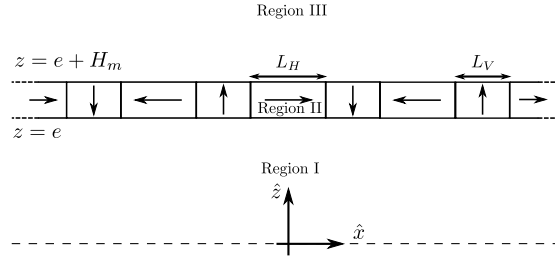


Fig. 3.2: Geometry for the magnetic field modelling. Magnet array is supposed to be periodic in the x direction and infinite in the y direction. The $z = 0$ plane is a symmetry as a consequence of the second magnet array.

The magnetic field is due to two different sources: the permanent magnets and the current in the electrical tracks. Neglecting the second source, the magnetic field and the beam are studied as two different problems. In addition, considering that the Lorentz force does not vary significantly with respect to the deflection of the beam, the beam problem can be studied as a linear problem with a fixed value of the Lorentz force, computed at rest state of the diaphragm.

3.1.1 Magnetic field modelling

To simplify the study of the magnetic field several simplifications are added:

- Magnets are supposed to be infinite in the y direction.
- The permanent magnet pattern is reproduced infinitely in the x direction.
- The magnetic permeability in magnets and in the airgap is supposed to be constant and equal to the one in vacuum.

According to the first hypothesis, the magnetic field can be studied as a 2D model. The geometry used for the model is the one shown in fig. 3.2. The approach used to solve the problem is the one presented in Meessen et al. (2010) or Gysen et al. (2010). Since only the magnetic field due to the permanent magnets is considered, equations are reduced to:

$$\nabla \times \vec{H} = 0 \quad (3.1)$$

$$\nabla \cdot \vec{B} = 0. \quad (3.2)$$

The fact that the magnetic field $\vec{H}$ is irrotational means that it can be derived from a scalar magnetic potential ϕ

$$\vec{H} = -\nabla\phi. \quad (3.3)$$

Materials are supposed to be homogeneous and linear i.e. saturation is not taken into account. Thus, the constitutive equations linking the magnetic field and the magnetic flux density $\vec{B}$ are:

$$\vec{B}_i = \mu_0 \vec{H}_i \quad \text{for } i = 1, 3 \quad (3.4)$$

$$\vec{B}_2 = \mu_0 \vec{H}_2 + \vec{B}_r, \quad (3.5)$$

where $\vec{B}_r$ is the remanent flux density in the magnets. Using (3.1) to (3.5), equations involving scalar potential only are obtained:

$$\mu_0 \Delta\phi_i = 0 \quad \text{for } i = 1, 3 \quad (3.6)$$

$$\mu_0 \Delta\phi_2 = \nabla \cdot \vec{B}_r. \quad (3.7)$$

The solution to this equation in the different regions is obtained through series expansion and by separating the solution between a solution to the homogeneous equation and a particular solution to non homogeneous equation. By using a separation of variables on the scalar magnetic potential:

$$\phi_i = X_i(x)Z_i(z) \quad (3.8)$$

the solution to the homogeneous equation in each region is written as:

$$X_i(x) = \sum_{n=0}^{\infty} A_{1i,n} \sin(\lambda_{i,n}x) + A_{2i,n} \cos(\lambda_{i,n}x) \quad (3.9)$$

$$Z_i(z) = \sum_{n=0}^{\infty} A_{3i,n} \exp(-\lambda_{i,n}z) + A_{4i,n} \exp(\lambda_{i,n}z). \quad (3.10)$$

Before applying the boundary conditions, a particular solution to the equation in the region occupied by magnets has to be found. This is done by expanding the

remanent magnetic field components in Fourier series. Polarization of magnets is supposed homogeneous, so the remanent magnetic field is only a function of x and can be expressed as:

$$B_{rx} = \sum_{n=0}^{\infty} b_{1,n} \cos(\lambda_n x) \quad (3.11)$$

$$B_{rz} = \sum_{n=0}^{\infty} b_{2,n} \sin(\lambda_n x) \quad (3.12)$$

where B_{rx} and B_{rz} are respectively the x and z component of the remanent flux density. The coefficients are obtained classically, using the cosine function orthogonality and are given by:

$$\lambda_n = \frac{n\pi}{L_V + L_H} \quad (3.13)$$

$$b_{1,n} = -\frac{4B_r}{n\pi} \sin\left(\frac{n\pi}{2}\right) \cos(n\pi) \cos\left(\lambda_n \frac{L_V}{2}\right) \quad (3.14)$$

$$b_{2,n} = \frac{4B_r}{n\pi} \sin\left(\frac{n\pi}{2}\right) \cos(n\pi) \sin\left(\lambda_n \frac{L_V}{2}\right) \quad (3.15)$$

in which B_r is the value of the remanent flux density in a magnet. The particular solution of (3.7) with remanent flux density given by (3.11) and (3.12) is simply:

$$\phi_{2,p} = \frac{1}{\mu_0} \sum_{n=1}^{\infty} \frac{b_{1,n}}{\lambda_n} \sin(\lambda_n x) \quad (3.16)$$

Now boundary conditions can be applied to find the unknown coefficients in the homogenous solution. The periodicity of the Halbach array ensures the periodicity of the magnetic potential ϕ_i which gives:

$$\lambda_{i,n} = \lambda_n. \quad (3.17)$$

The second Halbach array which is face to face with the first one imposes a null normal component of the magnetic flux density at the boundary $z = 0$. This means that:

$$A_{31,n} = A_{41,n}. \quad (3.18)$$

On the other side of the array, the magnetic potential has to vanish when z goes to infinity. This leads to:

$$A_{43,n} = 0. \quad (3.19)$$

The remaining unknown coefficients are obtained by imposing the following continuity conditions at the interfaces between the three regions:

$$B_{1z}(x, e) = B_{2z}(x, e) \quad (3.20)$$

$$H_{1x}(x, e) = H_{2x}(x, e) \quad (3.21)$$

$$B_{2z}(x, e + H_m) = B_{3z}(x, e + H_m) \quad (3.22)$$

$$H_{2x}(x, e + H_m) = H_{3x}(x, e + H_m) \quad (3.23)$$

where e is the thickness of the airgap and H_m is the height of the magnets. These conditions can be rewritten using (3.3), (3.4) and (3.5):

$$-\mu_0 \frac{\partial \phi_1}{\partial z}(x, e) = -\mu_0 \frac{\partial \phi_2}{\partial z}(x, e) + B_{rz} \quad (3.24)$$

$$\frac{\partial \phi_1}{\partial x}(x, e) = \frac{\partial \phi_2}{\partial x}(x, e) \quad (3.25)$$

$$-\mu_0 \frac{\partial \phi_2}{\partial z}(x, e + H_m) + B_{rz} = -\mu_0 \frac{\partial \phi_3}{\partial z}(x, e + H_m) \quad (3.26)$$

$$\frac{\partial \phi_2}{\partial x}(x, e + H_m) = \frac{\partial \phi_3}{\partial x}(x, e + H_m) \quad (3.27)$$

Using the orthogonality of the sine and cosine functions and by combining the equations all the coefficients are found.

Since the aim of the magnetic model is to evaluate the electromagnetic body forces, the only region of interest for the study is the region I, i.e. the air gap in which the scalar magnetic potential is:

$$\phi_1(x, z) = \sum_{n=1}^{\infty} a_n \sin(\lambda_n x) \cosh(\lambda_n z) \quad (3.28)$$

where:

$$a_n = \frac{-4\sqrt{2}B_r}{\mu_0 \lambda_n n \pi} \left(e^{-\lambda_n e} - e^{-\lambda_n (e+H_m)} \right) \times \cos(n\pi) \sin\left(\frac{n\pi}{2}\right) \sin\left(\lambda_n \frac{L_V}{2} + \frac{\pi}{4}\right). \quad (3.29)$$

This expression can be further simplified considering the fact that $\sin(\frac{n\pi}{2})$ is equal to zero if n is pair. Then the magnetic potential will be given by:

$$\phi_1(x, z) = a_0 + \sum_{m=0}^{\infty} a_m \sin(\lambda_m x) \cosh(\lambda_m z) \quad (3.30)$$

where:

$$a_m = \frac{(-1)^{2m+2} 4\sqrt{2}B_r}{\mu_0 \lambda_m (2m+1)\pi} \left(e^{-\lambda_m e} - e^{-\lambda_m (e+H_m)} \right) \times \sin\left(\lambda_m \frac{L_V}{2} + \frac{\pi}{4}\right) \quad (3.31)$$

$$\lambda_m = \frac{(2m+1)\pi}{L_V + L_H}. \quad (3.32)$$

a_0 can be fixed arbitrarily to zero in this case, since it has no influence on the displacement of the membrane. As it will appear in the mechanical model, it is not necessary to derive explicitly the magnetic field expression.

3.1.2 Mechanical modelling

The mechanical model is developed using the laminated beam theory, developed earlier, with layers made of elastic materials only. The diaphragms are supposed to be made of two continuous layers i.e. electrical track segmentation is not taken into account. The beam is loaded by an equivalent line load due to the Lorentz forces, as explained in previous chapter, which appear in the electrical tracks subjected to a current density J ; these are simply given by:

$$\begin{aligned}\vec{f} &= \vec{J} \times \vec{B} \\ &= J\hat{y} \times \left(-\mu_0 \frac{\partial \phi}{\partial x} \hat{x} - \mu_0 \frac{\partial \phi}{\partial z} \hat{z} \right) \\ &= \mu_0 J \frac{\partial \phi}{\partial x} \hat{z} - \mu_0 J \frac{\partial \phi}{\partial z} \hat{x}.\end{aligned}\quad (3.33)$$

This body force has two component but only one is taken in consideration for bending of the beam. The $\hat{x}$ component is neglected. From this volume force an equivalent force per unit length is obtained by integration over the thickness of the electrical conductor and the width of the beam:

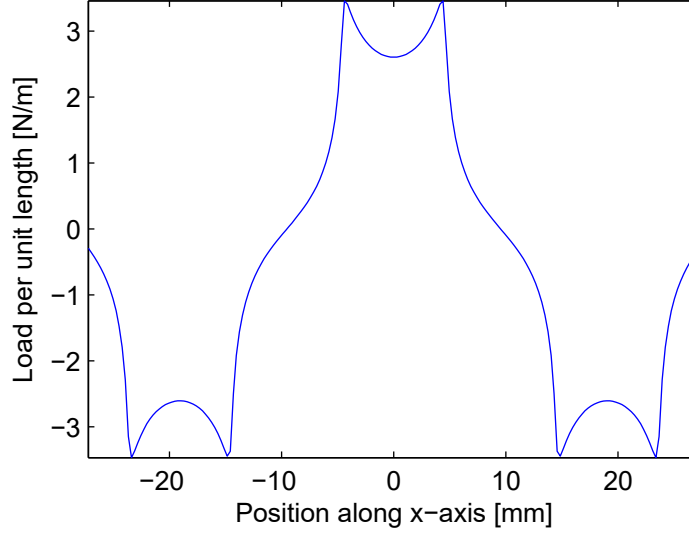
$$q(x) = w_b \mu_0 J \frac{d\Phi_0(x)}{dx} \quad (3.34)$$

where:

$$\Phi_0(x) = \int_{h_k}^{h_c} \phi(x, z) dz \quad (3.35)$$

where h_c and h_k are the coordinates in z of the upper and lower boundaries of the electric path, respectively, and w_b is the width of the beam. The equivalent load generated by the electromagnetic body forces is shown at fig. 3.3.

The aim here is to obtain an expression for the internal bending moment, which will be used to compute the curvature of the beam, as explained in previous chapter, which is given by (2.12). Considering the geometry of the beam showed in fig. 3.1. The M_b moment is applied to ensure the zero slope condition at $x = 0$ which is due to the symmetry of the problem. Moreover a force F_1 is applied at $x = L/2$, where L is the bubble length. It represents the reaction force of the second membrane. Moreover no bending moment is applied at $x = L/2$ since the curvature for $x > L/2$ is zero. The problem can seem over-constrained since three boundary conditions are imposed at $x = L/2$ i.e null displacement, null slope and null bending moment. In fact it is not since the length of the bubble L is unknown and part of the problem.

Fig. 3.3: Equivalent load q

Performing a static equilibrium of moments and forces on the beam gives the value of both unknowns F_1 and M_b :

$$F_1 = -w_b \mu_0 J \left(\Phi_0 \left(\frac{L}{2} \right) - \Phi_0(0) \right) - \frac{w_b L}{2} P_f \quad (3.36)$$

$$M_b = w_b \mu_0 J \left(\Phi_1 \left(\frac{L}{2} \right) - \frac{L}{2} \Phi_0(0) \right) + \frac{w_b L^2}{8} P_f \quad (3.37)$$

where Φ_i is computed from (3.35) by the following formula:

$$\Phi_i(x) = \int_0^x \Phi_{i-1}(x') dx'. \quad (3.38)$$

The internal bending moment is given by:

$$M(x) = -M_b + w_b \mu_0 J (\Phi_1(x) - x \Phi_0(0)) + w_b P_f \frac{x^2}{2} \quad (3.39)$$

and is used in the expression of the curvature (2.12), developed in the previous chapter. The slope of the beam is obtained by integrating this equation once. Because of the symmetry of the problem the slope at $x = 0$ has to be null. The general expression for the slope is then:

$$\frac{dv}{dx} = \frac{-M_b}{EI} x + \frac{w_b \mu_0 J}{EI} \left(\Phi_2(x) - \frac{x^2}{2} \Phi_0(0) \right) + \frac{w_b P_f x^3}{6EI}, \quad (3.40)$$

where EI is the equivalent bending stiffness given by $A_1/(w_b(A_2^2 - A_3A_1))$. The first root of this equation, evaluated in $L/2$, will give the value of half of the bubble length L . Due to the expression of $\Phi_2(x)$ this equation is implicit with respect to L and the roots will be determined numerically. The displacement is then obtained by integrating (3.40):

$$v(x) = \frac{-M_b}{EI} \frac{x^2}{2} + \frac{w_b\mu_0 J}{EI} \left(\Phi_3(x) - \frac{x^3}{6} \Phi_0(0) \right) + \frac{w_b P_f x^4}{24EI} + C_2 \quad (3.41)$$

where C_2 is given by imposing a null displacement at $x = L/2$. The volume per unit width of the micropump can then be evaluated by integrating (3.41) and multiplying it by four due to the double symmetry. It is given by:

$$V = \frac{-M_b}{EI} \frac{L^3}{12} + \frac{w_b\mu_0 J}{EI} \left(4\Phi_4\left(\frac{L}{2}\right) - \frac{L^4}{96} \Phi_0(0) \right) + \frac{w_b P_f L^4}{96EI} + 2C_2 L. \quad (3.42)$$

3.1.3 Finite element verification

In order to verify some of the assumptions, the analytical model is compared to 2D finite element simulations performed in COMSOL Multiphysics[®] software. These models do not consider the coordinate along the width of the beam. A first simulation studies the effect of the variation of the body force with respect to the displacement of the diaphragm. The segmentation of the electrical path, which means the insulating gap between each track, is not taken into account and the impact of the current density on the magnetic field is not studied either. In a second simulation, the segmentation is taken into account.

The geometry of the Comsol model is shown in fig. 3.4, where the domain 1 represents the surrounding environment and is modelled by air. Domains 2 to 18 represent the permanent magnets, which are modelled by imposing a remanent flux density of 1.25 T, respecting the Halbach array structure. Domains 19 to 26 are the electrical tracks, made in copper and domain 27 is made of kapton. Magnetostatic is used and thus the equations solved by Comsol are given by:

$$\nabla \times \mathbf{H} = \mathbf{J}_e \quad (3.43)$$

$$\nabla \times \mathbf{A} = \mathbf{B}, \quad (3.44)$$

where $\mathbf{H}$ is the magnetic field, $\mathbf{J}_e$ is the current density, which is null in this study, $\mathbf{A}$ is the potential vector and $\mathbf{B}$ is the flux density. Magnetic insulation is used as boundary condition for the outer boundaries of the domain. For the study of the diaphragm deflection, the equations used are the static equilibrium given by:

$$-\nabla \cdot \boldsymbol{\sigma} = \mathbf{F}_v \quad (3.45)$$

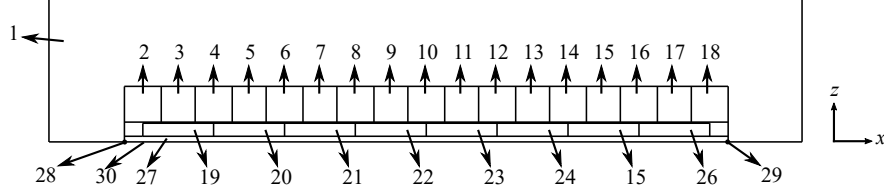


Fig. 3.4: Geometry of Comsol model. Domain 1 represents the surrounding environment and is modelled by air. Domains 2 to 18 represent the permanent magnets, which are modelled by imposing a remanent flux density. Domains 19 to 26 are the electrical tracks, made in copper and domain 27 is made of kapton. Domains 19 to 27 represent the diaphragm and have been magnified 10 times in the thickness direction.

where σ is the stress tensor and F_v is the volume force density. The displacement is fully constrained at point 28 whereas only its x component is constrained for point 29. Domain 21, 22 and 23 are submitted to a volume load given by the Lorentz force, where both x and z component are implemented, with a current density of $5 \text{ A}^2 \text{ mm}^{-1}$. The contact, which occurs between the two diaphragms at $z = 0$, is implemented by a counter reaction imposed at edge 30 if the diaphragm tries to go below $z = 0$. A spring like force in z direction is used and is given by:

$$F_A = k_s * |w| (w < 0), \quad (3.46)$$

where k_s is the spring stiffness and is fixed to $5 \times 10^8 \text{ N m}^{-3}$. A mesh made of triangular element is used and the study is performed in two steps. Both steps are stationary ones, where only the magnetic field is solved in the first step and only the displacement of the diaphragm in the second one. Both are solved with the solving strategy proposed by default.

The comparison of the computed displacement is shown in fig. 3.5. A good agreement is noticed between the three models and valid the considered assumptions. Even if the segmentation of the electrical tracks modifies the stiffness of the beam, the simulation shows that the impact is limited and can be neglected, at least in static analysis.

3.2 Piezoelectric micropump

In the piezoelectric micropump, the bending is ensured by the piezoelectric benders that are fixed to the elastic diaphragm. Sealing is ensured by the external pressure, applied on top of the patches. It will be modelled considering that the diaphragm and the patch are equivalent to a composite beam. Thus, equations developed previously

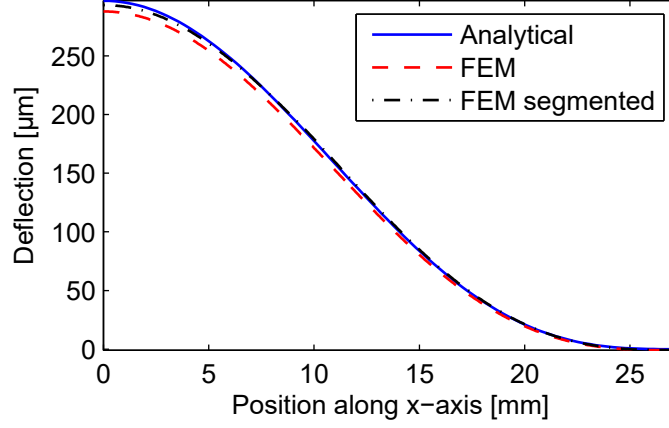


Fig. 3.5: Displacement comparison between the analytical model and finite element simulation in COMSOL Multiphysics[®]. A good agreement is noticed emphasizing the weak effect of electrical path segmentation and body force variation with displacement.

in chapter 2 will be used to compute the deformation of the diaphragm as well as the stresses and the electric field. Dynamics are completely neglected here and the fluid is only taken into account by assuming that it applies a known constant pressure on the diaphragm. Since the applied external pressure evolves linearly between the inlet and the outlet, they are no more longitudinal symmetry and the bubble has to be studied entirely. The second diaphragm is supposed to deform in a similar way so only the upper one is studied. The lower one is taken into consideration in the contact constraint only by applying meaningful boundary conditions i.e. a null displacement, a null slope and a null bending moment, at both ends of the bubble. The system is not over-constrained since the bubble length is also an unknown of the problem. The simplified problem is shown in fig. 3.6. Notes that L_1 and L_2 are both unknown. The length of a piezoelectric electrode is λ .

The simplifications introduced allow a complete analytical solution to the problem. The beam bends under the action of the mechanical load and of the piezoelectric effect. The bubble length is supposed to be larger than the zone which is powered i.e. larger than 3λ . Since the curvature is discontinuous at the transition between powered and unpowered sections the beam is divided in three parts:

- From $0 \leq x \leq L_1$, the beam is under mechanical load only and the curvature is given by (2.12).
- From $L_1 \leq x \leq L_1 + 3\lambda$, the beam is under mechanical load but three piezo-

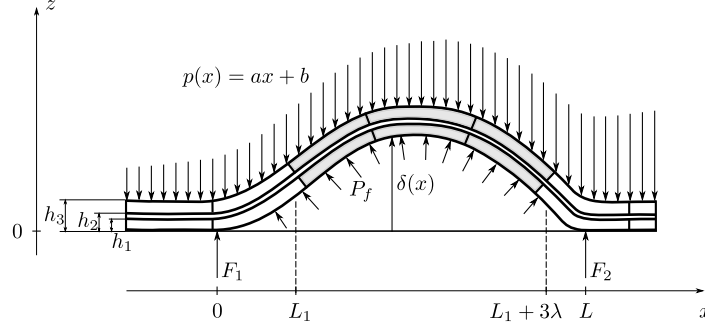


Fig. 3.6: Simplified geometry for modelling. Notes that L_1 and L_2 are both unknown. The length of a piezoelectric electrode is λ .

electric sections are powered too so in addition to the (2.12), (2.17) is added.

- From $L_1 + 3\lambda \leq x \leq L$, the curvature is once more only due to the mechanical load and thus given by (2.12).

According to static equilibrium with respect to fig. 3.6, the reaction forces F_1 and F_2 are given by:

$$F_1 = w_b \frac{L}{2} \left(a \frac{L}{3} + b - P_f \right) \quad (3.47)$$

$$F_2 = w_b \frac{L}{2} \left(a \frac{2L}{3} + b - P_f \right) \quad (3.48)$$

The internal bending moment is obtained by static equilibrium on a part of the beam and is given by:

$$M_b(x) = w_b \left(a \frac{L^2}{6} x + \frac{b - P_f}{2} x (L - x) \right). \quad (3.49)$$

So by integrating the curvature, the slope is obtained for the three sections. In addition, the null slope condition is imposed at both ends as well as its continuity between each section. This gives for $0 \leq x \leq L_1$

$$\delta'_l(x) = \frac{ax^4}{24I_E} - \frac{aL^2x^2}{12I_E} - \frac{b - P_f}{2I_E} \left(\frac{Lx^2}{2} - \frac{x^3}{3} \right) \quad (3.50)$$

for $L_1 \leq x \leq L_1 + 3\lambda$,

$$\delta'_c(x) = \frac{1}{\kappa_\nu} (x - L_1) + \delta'_l(x) \quad (3.51)$$

and for $L_1 + 3\lambda \leq x \leq L$,

$$\delta_r'(x) = \frac{3\lambda}{\kappa_\nu} + \delta_l'(x) \quad (3.52)$$

where I_E is given by

$$I_E = \frac{A_2^2 - A_3 A_1}{A_1}.$$

Using the last slope equation and imposing the null condition at L , a relation is found for the length of the bubble:

$$P_f - b = \frac{36\lambda}{\kappa_\nu} \frac{I_E}{L^3} + \frac{aL}{2}. \quad (3.53)$$

Integrating the slopes' expressions, an expression for the displacement $\delta(x)$ is obtained for $0 \leq x \leq L_1$:

$$\delta_l(x) = \frac{ax^5}{120I_E} - \frac{aL^2x^3}{36I_E} - \frac{b - P_f}{2I_E} \left(\frac{Lx^3}{6} - \frac{x^4}{12} \right) \quad (3.54)$$

for $L_1 \leq x \leq L_1 + 3\lambda$,

$$\delta_c(x) = \frac{1}{\kappa_\nu} \frac{(x - L_1)^2}{2} + \delta_l(x) \quad (3.55)$$

and for $L_1 + 3\lambda \leq x \leq L$

$$\delta_r(x) = \frac{3\lambda}{\kappa_\nu} \left(x - \left(L_1 + \frac{3\lambda}{2} \right) \right) + \delta_l(x). \quad (3.56)$$

By imposing a null displacement at the end of the beam, an expression for L_1 is obtained:

$$L_1 = \frac{L}{2} - \frac{3\lambda}{2} + \frac{1}{I_E} \frac{\kappa_\nu}{\lambda} \frac{L^5}{2160}. \quad (3.57)$$

The volume pumped V_f can be computed by integrating the displacement between the ends of the bubble and multiplying it by the width of the pump:

$$V_f = w_b \left(\left(\frac{a}{I_E} \right)^2 \frac{\kappa_\nu}{\lambda} \frac{L^{10}}{3110400} - \frac{3\lambda}{40\kappa_\nu} L^2 + \frac{9}{8} \frac{\lambda^3}{\kappa_\nu} \right). \quad (3.58)$$

3.2.1 Finite element verification

In this section, the displacement, the stresses and the electric field analytical expressions are compared to a 2D finite element model implemented in Comsol, in case of plane stress or plane strain hypothesis. Since both models give quite different results, a 3D model is used to see which hypothesis is the more suitable in the specific case of a 32 mm long, 6.4 mm wide and 0.66 mm high pumping structure.

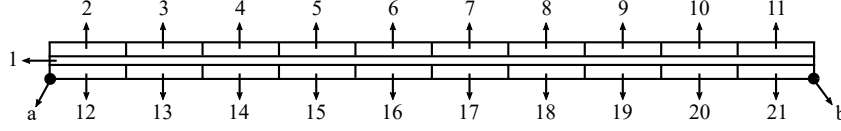


Fig. 3.7: Geometry for 2D Comsol model. Domain one is an elastic layer made of brass. Domains 2 to 11 are made of PZT-5H with polarization going up and domains 12 to 21 are made of PZT-5H with polarization going down.

The considered structure has two PZT-5H layers with opposite polarization which sandwiched a brass layer divided in 10 phases. It is expected that the plane strain hypothesis, corresponding to the null yy -strain in the developed theory for beams, has to be closer to the 3D FEM, since the width over thickness ratio is nearly 10.

2D finite element model

The geometry for the Comsol model is shown at fig. 3.7. On top of the structure, a pressure which varies linearly from 0 Pa to 50 kPa is imposed. At the opposite boundary, a boundary load is imposed to represent the contact condition with the bottom of the pump. It is imposed exactly like for the electromagnetic pump model with a spring stiffness k_s of $9 \times 10^{12} \text{ N m}^{-3}$. On the upper boundary of domains 6, 7 and 8, a voltage of 150 V is imposed whereas at their lower boundary and at the upper one of domain 16, 17 and 18, a voltage of 75 V is imposed. All other boundaries are electrically grounded. Domains 2 to 11 are made of PZT-5H with polarization going up and domains 12 to 21 are made of PZT-5H with polarization going down. They are all modelled using linear piezoelectricity and linear elasticity is used for the domain 1, made of brass. Displacement at point a is fully constrained whereas at point b, only the z component is constrained. A triangular mesh is used and a stationary study is performed, solved using the default solving strategy proposed by Comsol.

3D finite element model

The 3D model is just an extrusion of the 2D one and is shown in fig. 3.8. Only the boundary conditions for the displacement are changed and are now: fully constrained displacement at point a, constrained displacement in x and z direction for point b and constrained displacement in z direction only for point c.

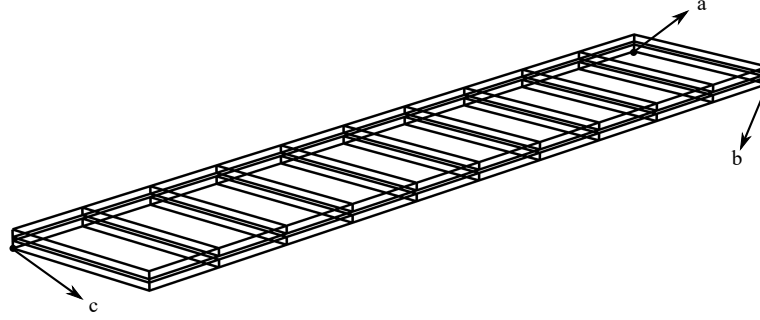


Fig. 3.8: Geometry for 3D Comsol model. It is simply an extrusion of the 2D model.

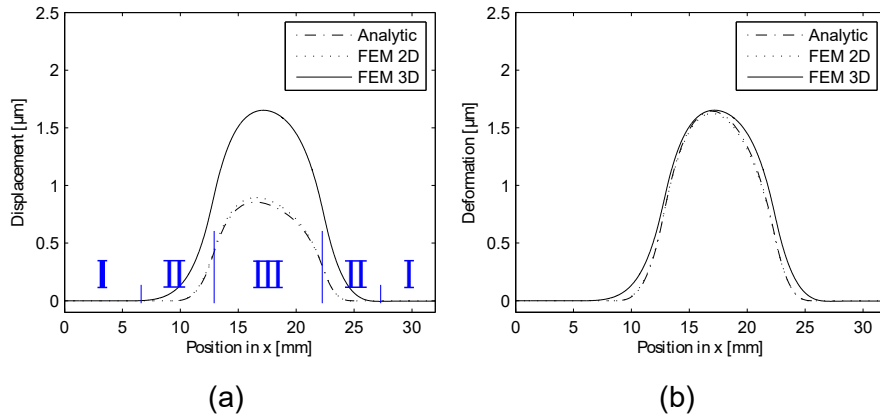


Fig. 3.9: Displacement in (a) null yy -stress and (b) null yy -strain assumption taken at half width.

Comparison

Figs. 3.9 (a) and (b) show the displacement in the case of a null yy -stress and null yy -strain assumptions, respectively. It is taken at half of the pump thickness and width in the brass layer. It appears clearly that the analytical models are in good agreement with the 2D FEM in both assumptions but, as expected, the null yy -strain is better suited to model what occurs in this particular geometry as shown by the 3D FEM. In both cases, the length of the bubble is slightly underestimated compared to the 3D model.

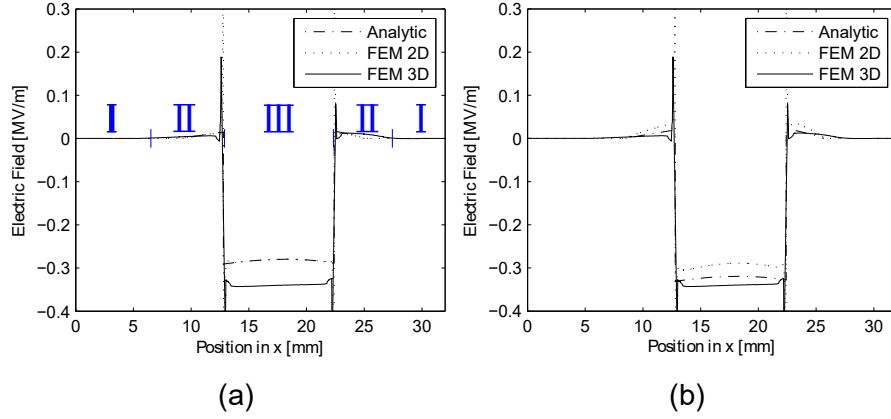


Fig. 3.10: Electric field at $z = h_1$ in the (a) null yy -stress and (b) null yy -strain assumption at half width.

Fig. 3.10 (a) and (b) present the electric field along x -axis at $z = h_1$ and at half-width for null yy -stress and null y -strain respectively. Three zones can be identified. In zone I the electric field is null. It is the case when no deformation occurs. In zone II the electric field is weak but non-zero. The piezoelectric segment are submitted to a zero electric potential but the bending leads to a non-zero electric field. In zone III the phases are actuated with an electric potential. In each figures, high peak can be observed at the interface between zones II and III. They are due to numerical problems in the finite element model which can be reduced by increasing mesh density. The conclusions for the electric field at $z = h_3$ shown in fig. 3.11 (a) and (b) are the same.

Fig. 3.12 (a) and (b) show the stress T_{xx} for null yy -stress or strain respectively at $z = h_1$ and at half-width. As for the electric field three zones can be identified. For both hypothesis the analytical model and the 2D FEM are in good agreements but not with the 3D FEM especially for the null yy -stress hypothesis where the stresses in the actuated part is nearly the double. Similar observations can be made from fig. 3.13 and 3.14 for the stresses at $z = h_2$ and at $z = h_3$ for both assumptions except that in the last case the null yy -stress assumption seems to be closer than in the null yy -strain hypothesis. This hypothesis is strengthened by fig. 3.15 where the stress T_{yy} is in quite good agreement between 2D and 3D for (a) at $z = h_1$ but where it is not the case for (b) and (c) where the stress has the opposite sign.

3.2.2 Limits of the one dimensional modelling

As was shown, the one dimensional model is not so bad in most of the cases for null yy -strain assumption, except for the null yy -stress. We attribute the latter to the fact

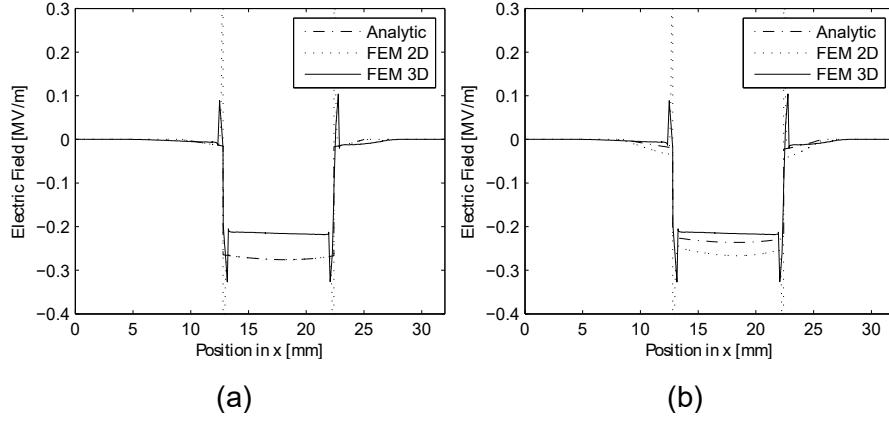


Fig. 3.11: Electric field at $z = h_3$ in the (a) null yy -stress and (b) null yy -strain assumption at half width.

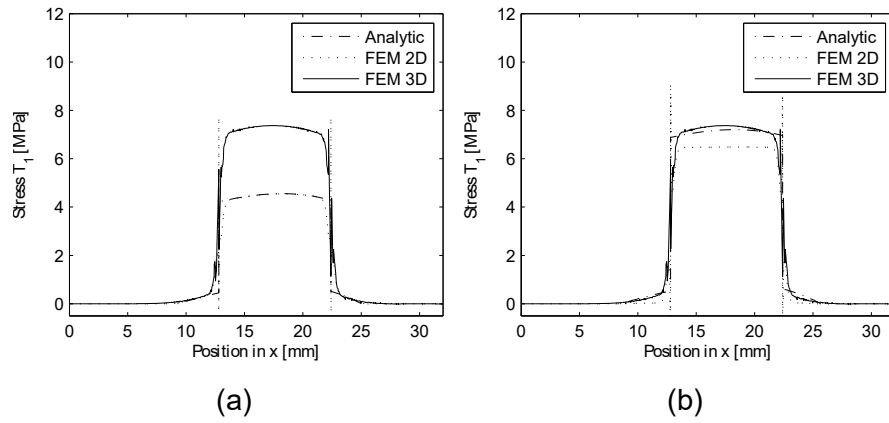


Fig. 3.12: Stress T_{xx} along x -axis at $z = h_1$ in the (a) null yy -stress and (b) null yy -strain assumption at half width.

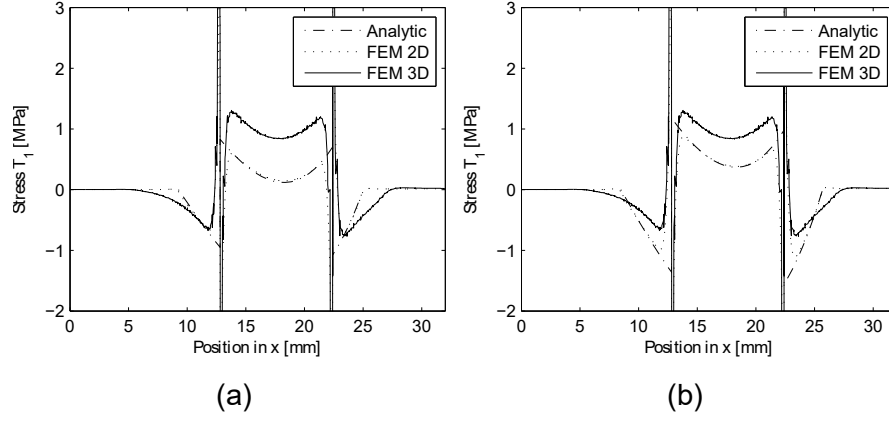


Fig. 3.13: Stress T_{xx} along x -axis at $z = h_2$ in the (a) null yy -stress and (b) null yy -strain assumption at half width.

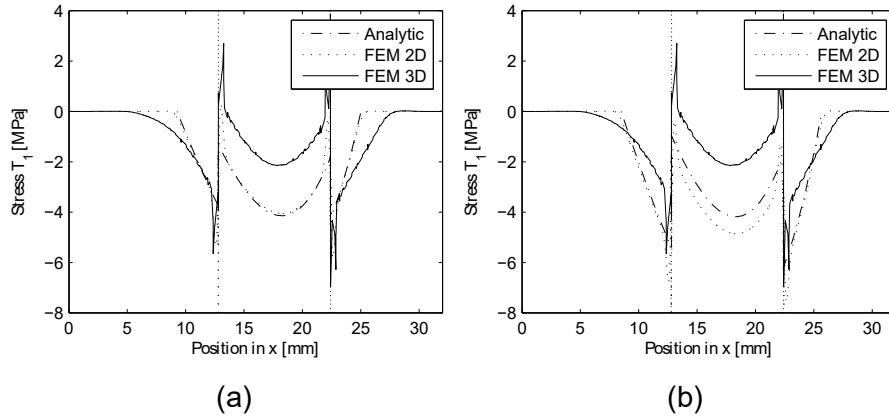


Fig. 3.14: Stress T_{xx} along x -axis at $z = h_3$ in the (a) null yy -stress and (b) null yy -strain assumption at half width.

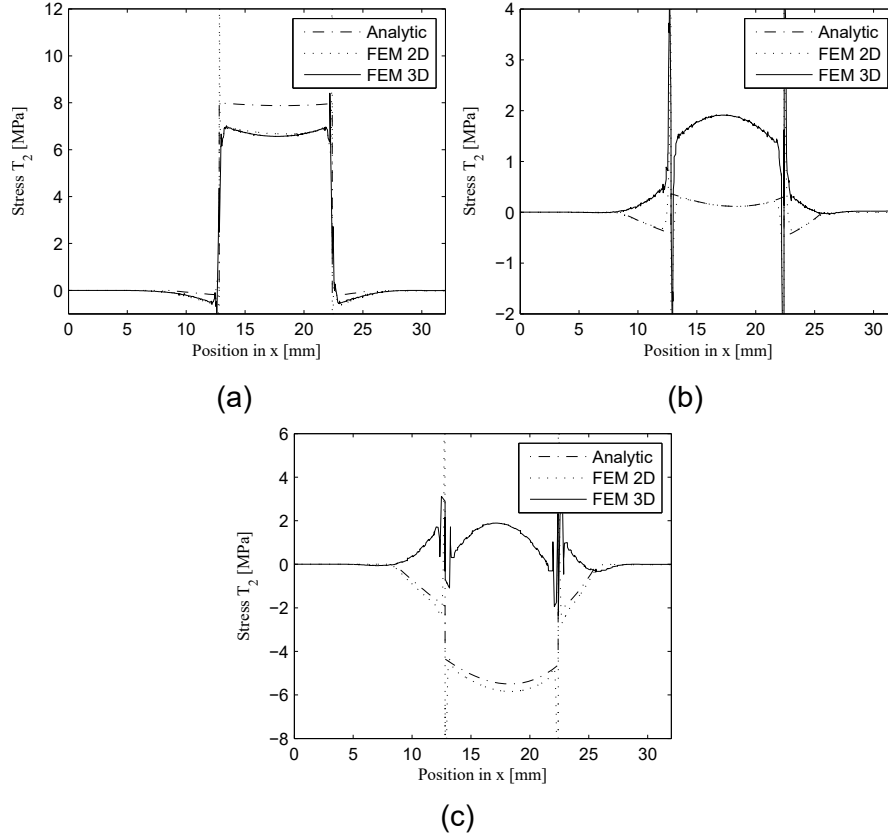


Fig. 3.15: Stress T_{yy} along x -axis at (a) $z = h_1$, (b) $z = h_2$ and (c) $z = h_3$ at half width. The high peaks oscillations at $x = 12.5$ and $x = 22.5$ are interface problems and are unphysical

that the model reaches its limits. As can be seen in fig. 3.17, the bubble starts to close in y direction and thus, the counter reaction which applies when there is a contact, loads the diaphragm, modify its deformations and its stress state.

To understand what occurs, let us vary the applied force density. This force density increase linearly along the length of the micropump from 0 to a maximum value. This value is changed and the following results are observed. At small value, the behaviour of the micropump follows the one of the beam, the effect of the yy curvature due to the piezoelectric effect is visible but its impact remains small, as shown in fig. 3.16. At higher value, such as in fig. 3.17, the boundary condition along the two long sides are no longer a free boundary condition but either simply posed or even clamped boundary conditions. This strongly influences the stress state in the mate-

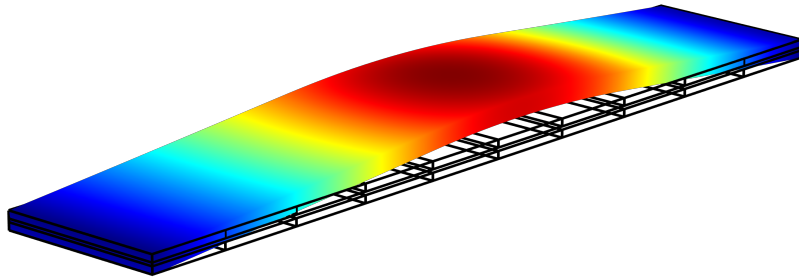


Fig. 3.16: 3D finite element model. The pump deflects in a beam fashion and the load does not vary with the y coordinates. In addition, the yy piezoelectric curvature, does not have a high impact on the deflection.

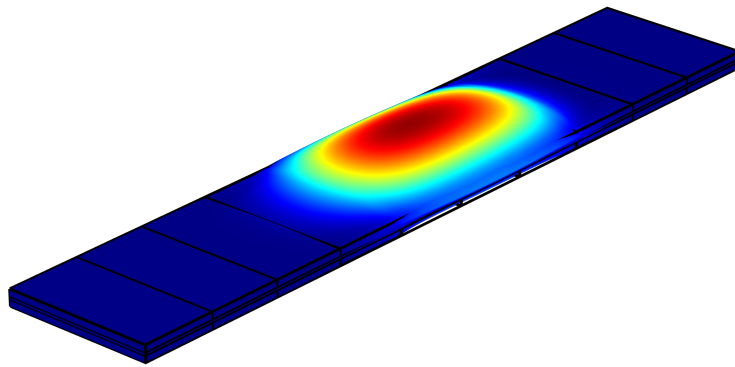


Fig. 3.17: 3D finite element model: the effect of the yy piezoelectric curvature on bubble shape is clearly visible. In addition, the high pressure applied lead to the contact of the two long side of the pump.

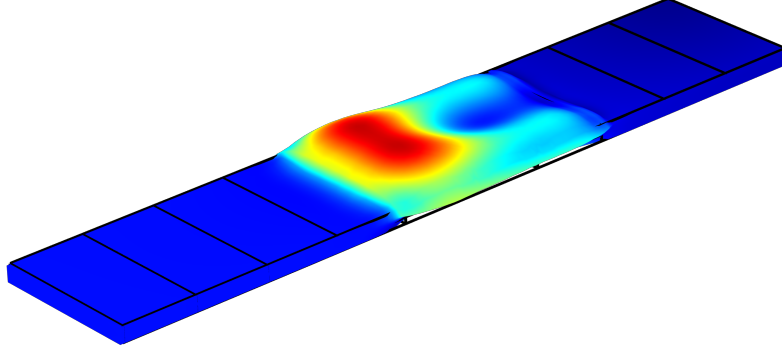


Fig. 3.18: 3D finite element model. The bubble shape is now strongly influenced the contact occurring in the y direction.

rial but also the volume of the bubble. For even higher load, as shown in fig. 3.18, the micropump can not be anymore studied by the beam theory and nothing can be predicted accurately. The shape of the bubble is due to the fact that a conformal contact between the diaphragm and the bottom of the micropump is not possible. Indeed, a conformal contact means that the diaphragm curvature should be null, meaning that the constant curvature imposed by the piezoelectric effect has to be exactly compensated by a constant bending moment, which can not be obtained by a constant force density. A more evolved theory able of taking into account loadings which vary in two dimension, such as a plate theory is essential.

3.3 Conclusion

In this chapter, equations to study the bending of laminated beam in static have been presented. Linear relations are considered, such that the effect of the different loading terms can be studied independently. Three different loads have been developed: a piezoelectric one, which uses the d_{31} coupling coefficient, a set of external pressure and forces and moment, and a Lorentz force density. The first uses the mean value of the electric field to compute the deflection of the beam, but integrates the effect of the bending curvature on the electric field, and through it, on the stress in the piezoelectric material. In the second case, the loads are used to compute an internal bending moment, $M(x)$, from which the curvature of the beam is expressed. The last load is similar to the previous one, since the Lorentz force density, which appears due to the interaction of a current density and of the magnetic flux density, is integrated over the layer to obtain an equivalent line load. This one is then used as in the second load case.

These equations have been used to compute the stress and electric fields in the case of a bender actuator, showing a gain in accuracy up to 40 percent compared to model where electric field is considered as constant over the thickness of the layer. It has then been applied to the micropump benchmark cases, for which displacement, stress and electric field are computed and compared to finite element model, made in Comsol multiphysics software which have also been described. They have shown their limits in evaluating the stress along the y axis in the piezoelectric micropump, showing that neither null yy -stress nor null yy -strain assumptions were suitable for its study. This has been further analysed using the 3D model made in Comsol and showed that in some circumstances, other effects should be taken into consideration, such as the contact appearing in the y direction, meaning a more evolved model able to consider two dimensional loads should be used.

Part II

Modelling - Two dimensional approach

Introduction

The previous chapter shows that the one dimensional theory, developed in chapter 2, is not sufficient to represent faithfully what occurs in the micropumps concepts. This is especially true for stress computation but also for displacement and volume computation in some cases. This is due to the fact that usual assumptions, i.e. null stress T_{yy} or null strain S_{yy} , do not represent properly the transverse curvature, due the d_{32} piezoelectric coupling coefficient. In addition, it does not incorporate what occurs in case of partial contact along the width coordinate, as shown in fig. 3.19 (a). This influences the displacement of the diaphragms and change the stress state in the material.

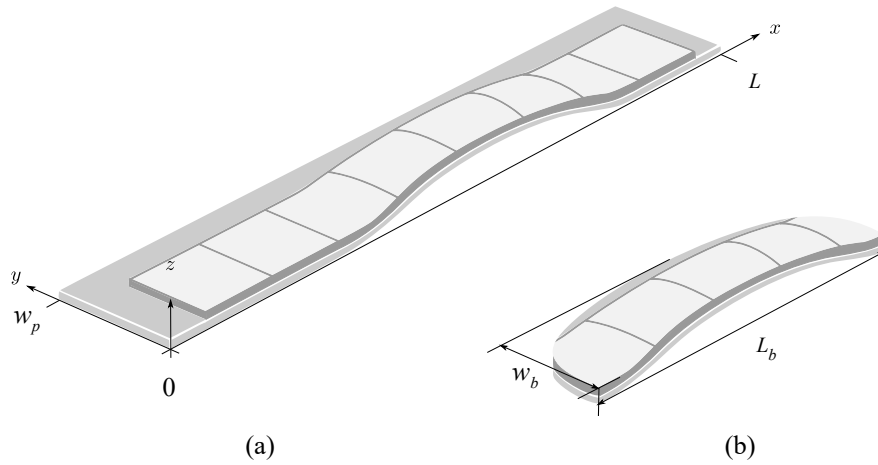


Fig. 3.19: (a) Diaphragm where edges $x = 0$, $x = L$ and $y = w_p$ are clamped. The edge $y = 0$ is a symmetry axis. (b) Equivalent plate, limited to the deformed structure, clamped along its contact line. Length L_b can not be considered as significantly bigger than the width w_b .

If literature is not so developed for one dimensional model of laminated structures which incorporate a piezoelectric material, it is certainly linked to the fact that they do not represent faithfully the underlying physics. This is a problem for sizing or optimization process where objective function will be the displacement or a derived quantity. In addition, exceeding the tensile yield stress or coercive electric field will destroy the device. Micropump concepts are example of such devices. Moreover, mechanical contact, that appears between the two diaphragms in the y direction and that considerably reduce the volume of the bubble, can not be taken into account neither.

Another remark can also be raised on the geometrical aspect ratio. A beam is normally a slender structure with two dimensions particularly smaller than the third one. If it is true that the diaphragms are quite long compared to their thickness and width, it is no longer the case if the bubble is thought as an independent structure, simply supported along its contact line ,as shown in fig. 3.19 (b). Indeed, the bubble has a length, L_b , which is comparable to its width, w_b , and a much smaller thickness. Hence the need for a more evolved model.

This second part explains the two dimensional models considering the diaphragms as bending plates. Chapter 4 describes the modelling of laminated structures under the Kirchhoff-Love assumptions for plates theory, which is also known as the classical laminated plate theory. As in the beam case, each layer can be made of an elastic or piezoelectric material. Constitutive equations for their dynamics are developed and a numerical scheme is provided in appendix B to solve them. It is then applied in various case studies, which are compared to 3D models made in Comsol Multiphysics.

Chapter 5 explains how the contact is taken into account. This includes the physics, as well as the different assumptions made, but also the formulation, discretization and algorithm used to include it in the previously developed numerical scheme of chapter 4 and appendix B. After that, all these tools are applied the micropump concepts in chapter 6.

Constitutive equations for laminated Kirchhoff-Love plates

4

Structures with a dimension, usually called the thickness, significantly smaller than the two others are known in literature as plates. Many constitutive equations have been developed and reported in the literature. They differ by the underlying assumptions, the plates' geometry and the loading. This influences their range of validity and their computational cost, and thus, should be selected carefully.

In this chapter, a brief state of the art of the various plate models is firstly given, highlighting their features, their gain and their shortcomings, ending with the contributions of the model which will be developed in the following section. These constitutive equations for the dynamics of thin laminated plates are based on Kirchhoff-Love hypothesis, the counterpart of Euler-Bernoulli for two dimensions.

A weak form of the equations is then deduced using the virtual work principle. They are then transformed to a system of ordinary differential equations using finite element method for spatial discretization. This system is integrated in time applying the generalized alpha method. This numerical scheme is used to analyse the behaviour of a piezoelectric monomorph subjected to transient loads. Both actuation and sensing are evaluated and compared to a 3D FEM model made in Comsol Multiphysics.

4.1 Overview of literature

Looking at the literature, it appears that modelling laminated plate structures which include one or more piezoelectric layers has been the subject of a quite huge amount of papers. These have been the subject of some reviewing article, such as Saravanos and Heyliger (1999) and Kapuria et al. (2010). The first attempts to take into account

the piezoelectric layers were through effective forces and moments due to the piezoelectric material used as an actuator, a source of strain, and are known as *induced strain models*. They did not consider the full electromechanical coupling, neglecting the direct piezoelectric effect.

Then several theories based on constructing an equivalent single layer plate theory have been proposed. In these works, a single expression, for the whole thickness, is guessed for the displacement of the plate. Depending on the order of the polynomial expansion along the thickness coordinate z , different theories are obtained. These theories are in fact extension of theories used for composite plates to include direct and inverse piezoelectric effect. Using a first order expansion in z , the classical laminated theory or the first order shear deformation theory is obtained depending if shear deformation are considered or not. Third order theory, such as the one in Mitchell and Reddy (1995), and even higher order theories have been proposed according to Kapuria et al. (2010), augmenting the complexity and computational load. Compared to the previously cited theories, the electric potential is now considered as a state variable and also expressed as an expansion with respect to the thickness coordinate z . However, these studies do not consider the link between the electric field expansion and displacement expansion such as suggested by the constitutive equations of piezoelectric material and electrical displacement equilibrium. In addition, according to Kapuria et al. (2010), the global expansion do not allow to represent the slope discontinuity of in plane displacements and continuity of transverse shear stresses at layer interfaces, leading to inaccurate global and local response for moderately thick plates.

In response to that problem, layerwise theories have been proposed. In this case, each layer has its own displacement expansion and only continuity of the displacement is imposed between each layer. This however multiplies the number of displacement unknowns by the the number of layers. A recently published paper Plagianakos and Papadopoulos (2015), using high order layerwise description of the displacement, seems to go deeper in the analysis of the electric field expansion with respect to the thickness coordinate but keeps the electric potential as a general independent variable.

In order to keep the efficiency of the equivalent single layer theory, the number of variables is reduced by imposing the transverse shear continuity conditions at layer interfaces and shear traction free conditions at top and bottom surfaces. This is what is done in zigzag theories, according to Kapuria et al. (2010). These theories are however unable to predict correctly the transverse shear stress. This issue has been solved in Kapuria and Nath (2013) using global/local approach, but it involves 13 primary variables for displacement and electric potential.

Apart from the thickness expansion used to describe the displacement, other

characteristics can be used to differentiate the various theories. Most of developed theories are based on linear relations. Nonlinear effects can appear at two stages in the modelling. The first one is in cases where displacement can not be considered as small anymore and thus a nonlinear expression of the strain has to be used. It has been considered in Dash and Singh (2009) or Zhang and Schmidt (2014). Another source of nonlinearity is in the constitutive equations of piezoelectric materials where a nonlinear model should be used in case of high electric field or if some more advanced things wants to be taken into account such as hysteresis. In addition to electrical potential loads and mechanical loads, some theories also take into account loads due to thermal fields present in the laminates. Very few articles study the case of interaction with electromagnetic fields. This is the case in Kim et al. (2015), where the effect of electromagnetic and thermal fields are studied using first order shear deformation single layer theory.

In addition to the review, Kapuria et al. (2010) also gives some challenges which should be addressed in the incoming modelling efforts and which are:

- Development of efficient 2D laminate theories which accurately predict the interlaminar transverse shear and normal stresses, mainly because these are the predominant cause of failure.
- Efficient and robust finite element for plates and shells with electroded piezoelectric actuators or sensors.
- Development of Benchmark solutions for computation of free-edge stresses.
- Including nonlinear piezoelectric coupling.

In this thesis, an efficient and easy to implement theory which reasonably represent the underlying physics of thin laminated plate structures has been sought. That is why assumptions made in equivalent single layer theory known as classical laminated theory has been chosen. In addition, in order to reduce even more the computational load, the developed theory takes advantage of the equipotential boundary condition imposed by the electrodes and solves electrical equilibrium equation exactly with respect to the chosen thickness expansion for the displacement, neglecting edge effects. This gives a laminated plate theory which is expressed with respect to mechanical displacement but keeps the strong electromechanical coupling. In cases where the voltage is not known a priori, but rather depends of an external dynamics, such as in sensors applications, only one ordinary differential equations per piezoelectric segment has to be solved. This approach, which has not be observed in the previously cited literature, gives a strong mathematical basement for the justification of the electric potential expansion with respect to the thickness coordinate. Moreover, the approach is not restricted to the classical laminated plate theory but can

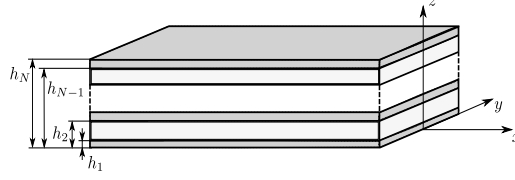


Fig. 4.1: Laminated plate and associated coordinate system.

be applied to more evolved theory presented in the literature. This is the first contribution. The second one is the numerical scheme proposed to solve them, based on finite element for space discretization and a generalized alpha method for time integration of mechanical equations. If the electrical equation has also to be solved, both system are solved in a partitioned, loosely coupled way, using a Runge-Kutta solver for the electrical equation.

4.2 Dynamics of thin laminated plates

The considered composite plates and its associated coordinate system is shown in fig. 4.1. All layers are supposed to be of constant thickness. Equations are obtained in a way similar to the one used for the beams. A guess on the form of the displacement field is made by making the following assumptions:

- Cross sections sections remains planar after deformation.
- The thickness of the plates remains constant.
- The normal to the midsurface remains normal after deformation
- The bonding between layers is perfect i.e. it ensures strains continuity.
- The displacement remains small.

These are known as the Kirchhoff-Love hypothesis and allow to express the displacement components in $\hat{x}$, $\hat{y}$ and $\hat{z}$ directions, respectively u , v , and w , as linear functions of the z coordinate. This can be deduced by simple geometrical considerations as shown in fig. 4.2. This leads to the introduction of two other function u_0 and v_0 which, with w , are independent of the z variable and are given by:

$$u = u_0(x, y) - zw_{,x} \quad (4.1)$$

$$v = v_0(x, y) - zw_{,y} \quad (4.2)$$

$$w = w(x, y). \quad (4.3)$$

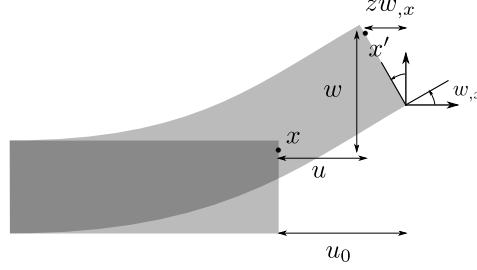


Fig. 4.2: Displacement geometry.

This is a classical expression for displacement and maybe the simplest. More advanced models, based on relaxation of some of the hypothesis or higher order expression for the approximation of u and v , have been developed such as reported in Reddy (2004). But, according to the author, it increases the computational cost considerably and should be used only if necessary.

4.2.1 Dynamic equilibrium

Before going further, let us define the sign convention for internal forces and moments. The plate is supposed to be in plane stress state which means only T_{xx} , T_{yy} and T_{xy} are non zero. Considering this assumption, the force density acting on an infinitesimal parallelepiped is simplified. For a facet with normal $\hat{x}$, this force density vector $\mathbf{t}$ is given by:

$$\mathbf{t} = \mathbf{T}^T \cdot \hat{x} \quad (4.4)$$

$$= T_{xx}\hat{x} + T_{xy}\hat{y}. \quad (4.5)$$

It is linked to the internal forces by simple integration over the thickness:

$$\begin{aligned} \mathbf{N} &= \int_0^h \mathbf{t} dz \\ &= N_{xx}\hat{x} + N_{xy}\hat{y}. \end{aligned} \quad (4.6)$$

In a similar way the internal moments can be computed using the cross product of the force density. For the considered facet this gives:

$$\begin{aligned} \mathbf{M} &= \int_0^h z \hat{z} \times \mathbf{t} dz \\ &= M_{xx}\hat{y} - M_{xy}\hat{x} \end{aligned} \quad (4.7)$$

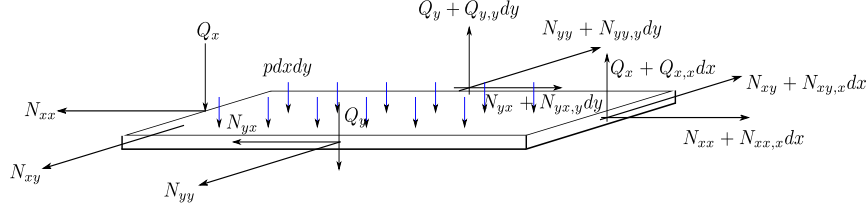


Fig. 4.3: Internal forces and external loads on an infinitesimal part of the plate.

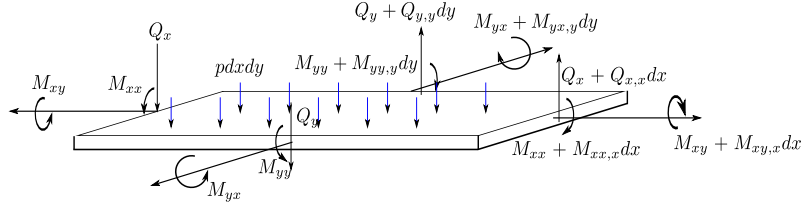


Fig. 4.4: Internal moments in an infinitesimal part of the plate.

where forces and moments are given by:

$$N_{ij} = \sum_{k=1}^N \int_{h_{k-1}}^{h_k} T_{ij} dz \quad (4.8)$$

$$M_{ij} = \sum_{k=1}^N \int_{h_{k-1}}^{h_k} z T_{ij} dz. \quad (4.9)$$

Using these notations, internal forces and moments, as well as their signs, are shown in fig.4.3 and fig. 4.4.

By applying the Newton's law to an infinitesimal part of the plate a set of relations are deduced linking internal forces and moments to the different accelerations. Computing the sum of internal and external forces in x direction a first relation is obtained:

$$\begin{aligned} dxdy \int_0^{h_N} \rho(z) \ddot{u}(z) dz &= (N_{xx} + N_{xx,x}dx) dy - N_{xx}dy \\ &+ (N_{yx} + N_{yx,y}dy) dx - N_{yx}dx \end{aligned} \quad (4.10)$$

where ρ is the mass density and $(\cdot)_{,i}$ stands for the partial derivative of $(\cdot)$ with respect to i . The integrals in left hand side of (4.10) can be expressed easily in function of u_0 , v_0 and w by defining some equivalent inertia terms:

$$I^k = \sum_{i=1}^N \frac{1}{k} \rho_i (h_i^k - h_{i-1}^k) \quad (4.11)$$

Notes that $\ddot{u}(z)$ in (4.10) is the second time derivative of 4.1. After simplification, this gives:

$$N_{xx,x} + N_{yx,y} = I^1 \ddot{u}_0 - I^2 \ddot{w}_{,x} \quad (4.12)$$

Doing the same for y and z axis, two more relations are obtained:

$$N_{xy,x} + N_{yy,y} = I^1 \ddot{v}_0 - I^2 \ddot{w}_{,y} \quad (4.13)$$

$$Q_{x,x} + Q_{y,y} = p + I^1 \ddot{w} \quad (4.14)$$

and two equations for moment equilibrium around $\hat{y}$ and $\hat{x}$ axis, respectively:

$$M_{xx,x} + M_{yx,y} - Q_x = I^2 \ddot{u}_0 - I^3 \ddot{w}_{,x} \quad (4.15)$$

$$-M_{yy,y} - M_{xy,x} + Q_y = -I^2 \ddot{v}_0 + I^3 \ddot{w}_{,y} \quad (4.16)$$

In addition, the internal loads have to be linked to the displacement field. This can be done by considering the constitutive equations of the materials, linking the strains $\mathbf{S}$ to the stress field $\mathbf{T}$.

4.2.2 Stress-strain relations

The stress field will depend of the layer. Considering linear piezoelectricity and linear stress-strain relation, each layer is described by the following law:

$$\begin{pmatrix} \mathbf{T} \\ \mathbf{D} \end{pmatrix} = \begin{pmatrix} \mathbf{c}^E & -\mathbf{e}^T \\ \mathbf{e} & \boldsymbol{\epsilon}^S \end{pmatrix} \begin{pmatrix} \mathbf{S} \\ \mathbf{E} \end{pmatrix} \quad (4.17)$$

where $\mathbf{c}^E$ is the material stiffness matrix, $\mathbf{e}$ is the piezoelectric coupling coefficient matrix, $\boldsymbol{\epsilon}^S$, the material permittivity matrix, $\mathbf{D}$, the electric displacement field, $\mathbf{S}$, the strains and $\mathbf{E}$, the electric field. Note that each layer can be considered as a piezoelectric one where, for purely elastic layers, $\mathbf{e}$ is simply the null matrix and the electric and electric displacement fields, since they are not coupled to the mechanical equations, are useless. The strain tensor is given by:

$$\mathbf{S} = \frac{1}{2} (\nabla \mathbf{u} + \nabla \mathbf{u}^T). \quad (4.18)$$

Piezoelectric materials are usually transversely isotropic with respect to their polarization direction. This means that $c_{11} = c_{22}$, $c_{12} = c_{21}$, $c_{13} = c_{31} = c_{23} = c_{32}$ and $e_{31} = e_{32}$. In addition the plate is supposed to be in a plane stress state which means only T_{xx} , T_{yy} , T_{xy} , S_{xx} , S_{yy} , S_{xy} and S_{zz} are non zero. Considering electrical variables, each layer is of constant thickness and very thin, which means, if side effects are neglected, that the only significant component of the electric displacement and electric field will be the z component. Taking into consideration all these

assumptions leads to:

$$\begin{pmatrix} T_{xx} \\ T_{yy} \\ 0 \\ T_{xy} \end{pmatrix} = \begin{pmatrix} c_{11} & c_{12} & c_{13} & 0 \\ c_{12} & c_{11} & c_{13} & 0 \\ c_{13} & c_{13} & c_{33} & 0 \\ 0 & 0 & 0 & c_{66} \end{pmatrix} \begin{pmatrix} S_{xx} \\ S_{yy} \\ S_{zz} \\ S_{xy} \end{pmatrix} + \begin{pmatrix} e_{31} \\ e_{31} \\ e_{33} \\ 0 \end{pmatrix} E_z. \quad (4.19)$$

The third equation can be used to get rid off S_{zz} from the two first equations. This leads to the definition of corrected compliance coefficients, which are given by:

$$c_{11}^* = c_{11} - c_{13}^2/c_{33} \quad (4.20)$$

$$c_{12}^* = c_{12} - c_{13}^2/c_{33} \quad (4.21)$$

$$e_{31}^* = e_{31} - c_{13}e_{33}/c_{33}. \quad (4.22)$$

Using these starred coefficient, the stresses are now given by:

$$\begin{pmatrix} T_{xx} \\ T_{yy} \\ T_{xy} \end{pmatrix} = \begin{pmatrix} c_{11}^* & c_{12}^* & 0 \\ c_{12}^* & c_{11}^* & 0 \\ 0 & 0 & c_{66} \end{pmatrix} \begin{pmatrix} S_{xx} \\ S_{yy} \\ S_{xy} \end{pmatrix} + \begin{pmatrix} e_{31}^* \\ e_{31}^* \\ 0 \end{pmatrix} E_z \quad (4.23)$$

All the ingredients for the mathematical description of the mechanical part are now given. It remains to consider the electrical behaviour.

4.2.3 Electrical field study

Since piezoelectric materials in the applications are essentially dielectric materials bonded by two conductive electrodes, the electric displacement and electric field have to verify the following conservation rules in each piezoelectric layer:

$$\nabla \cdot \mathbf{D} = 0 \quad (4.24)$$

$$\nabla \times \mathbf{E} = 0 \quad (4.25)$$

$$(4.26)$$

The fact that the electric field is irrotational means it derives from a scalar potential. Moreover, as previously said, only the z component of both field is significant. This simplifies drastically the equations above:

$$D_{z,z} = 0 \quad (4.27)$$

$$E_z = -V_{,z} \quad (4.28)$$

Considering the piezoelectric coupling permits to link the electric field to the displacement w , and to the electric potential V , through an ordinary differential equation on z . The z component of the electric displacement is given by:

$$D_z = e_{31}S_{xx} + e_{32}S_{yy} + e_{33}S_{zz} + \epsilon_{33}^S E_z \quad (4.29)$$

$$= e_{31}^*(S_{xx} + S_{yy}) + \left(\epsilon_{33}^S + \frac{e_{33}^2}{C_{33}} \right) E_z. \quad (4.30)$$

where the third line of (4.19) has been used to get rid of the strain in z direction, S_{zz} , and where the star coefficients have been defined in (4.22). Using the condition (4.27) for the electric displacement and using the definition of the strain and displacement leads to:

$$D_{z,z} = -e_{31}^* (w_{,xx} + w_{,yy}) + \left(\epsilon_{33}^S + \frac{e_{33}^2}{C_{33}} \right) E_{z,z} \quad (4.31)$$

and after some rearrangements and considering the definition of e_{31}^* :

$$E_{z,z} = \frac{e_{31}c_{33} - e_{33}c_{31}}{e_{33}^2 + \epsilon_{33}^S c_{33}} \Delta_\pi w = \alpha \Delta_\pi w, \quad (4.32)$$

where Δ_π is the 2D in plane Laplacian. Considering that for the i th layer a voltage V_i is applied on the top electrode and V_{i-1} on the bottom one, the electric field is given by:

$$E_{zi} = \alpha_i \Delta_\pi w \left(z - \frac{h_i + h_{i-1}}{2} \right) - \frac{V_i - V_{i-1}}{h_i - h_{i-1}}. \quad (4.33)$$

This expression shows that the electric field is determined by the curvature of the plate, obtained through the in plane Laplacian of the displacement component w , and the applied voltage difference. This means that the electric field can be replaced in the equations and the applied voltage will be part of the plate loading. This is of particular interest if the applied voltage can be considered as applied externally as in actuator applications. In the case where the study has to take into account the dynamics of the electrical circuit, such as in sensing applications or for power electronic design, an additional equation has to be considered for the current.

It can be obtained considering the Gauss's law applied on a closed box of infinitesimal thickness, around the interface between the piezoelectric material and the electrode, as shown in fig. 4.5:

$$\int_{\partial\Omega} \mathbf{D} \cdot \hat{n} dA = Q \quad (4.34)$$

where Q is charge enclosed by the volume defined by $\partial\Omega$, $\hat{n}$ is the normal to the surface $\partial\Omega$, going outward. The law is applied to a piezoelectric layer covered by its electrodes as shown in fig. 4.5. The box for Gauss's law completely encloses the interface between the electrode and the piezoelectric layer. Since this one is made of a conductive material and that electric displacement is null in it at equilibrium, the electric charge at the interface between the electrode and the piezoelectric layer is given by:

$$Q = - \int_A D_z(h_i) dS. \quad (4.35)$$

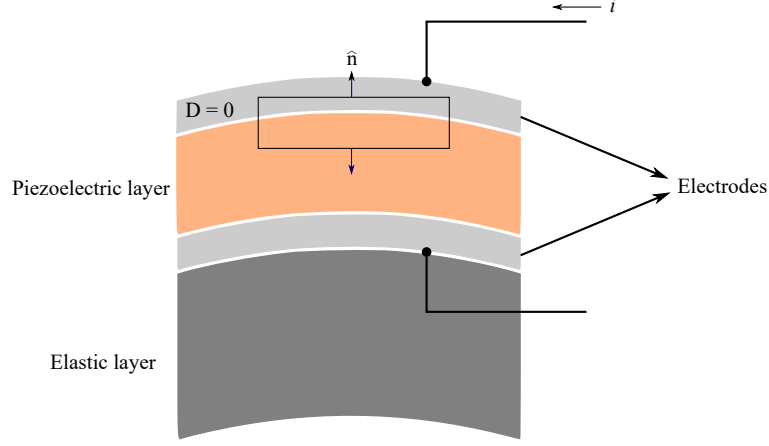


Fig. 4.5: Geometry to find the expression for electrical current. The black box is the one used to apply Gauss's law, where the arrows are the outward normal. The electrodes are in conductive materials and thus, the electric field as well as the electric displacement field are null. The enclosed charge is thus the one due to the piezoelectric layer.

where A is the surface of the electrode, D_z is the z component of the electric displacement field in the piezoelectric layer, evaluated at the upper electrode, Q is the electric charge at the upper electrode. By taking the time derivative of the equation above, an expression is obtained for the current. This can still be done in the electrostatic assumption since the time scales of electromagnetic phenomena and mechanical dynamics are quite different. This yields:

$$i_p(t) = -e_{31}^* \int_A \dot{u}_{0,x} + \dot{v}_{0,y} - h_m \Delta_\pi \dot{w} dS + Cp \frac{dV}{dt} \quad (4.36)$$

with

$$h_m = \frac{h_i + h_{i-1}}{2} \quad (4.37)$$

$$Cp = \frac{A}{h_i - h_{i-1}} \left(\epsilon_{33}^S + \frac{e_{33}^2}{c_{33}} \right) \quad (4.38)$$

where the change in the area of the electrode has been neglected and Δ_π is the in plane Laplace operator. This expression is valid for the current generated by a piezoelectric layer covered by an electrode. In case where two piezoelectric layers share an electrode, their effects have to be cumulated.

4.3 Summary of the set of equations

Internal loads, displacement and electric field can now be linked using (4.8), (4.9) (4.23) and (4.33):

$$\begin{pmatrix} N \\ M \end{pmatrix} = \begin{pmatrix} A^1 & -A^2 \\ A^2 & B - A^3 \end{pmatrix} \begin{pmatrix} S_{uv} \\ S_w \end{pmatrix} + \begin{pmatrix} \gamma_0 \\ \gamma_1 \end{pmatrix} \quad (4.39)$$

in which

$$N = (N_{xx}, N_{yy}, N_{xy})^T \quad (4.40)$$

$$M = (M_{xx}, M_{yy}, M_{xy})^T \quad (4.41)$$

$$S_{uv} = \left(u_{0,x}, v_{0,y}, \frac{1}{2} (u_{0,y} + v_{0,x}) \right)^T \quad (4.42)$$

$$S_w = (w_{,xx}, w_{,yy}, w_{,xy})^T \quad (4.43)$$

$$A^k = \sum_{n=1}^N \begin{pmatrix} c_{11}^* & c_{12}^* & 0 \\ c_{12}^* & c_{11}^* & 0 \\ 0 & 0 & c_{66} \end{pmatrix}_n \frac{h_n^k - h_{n-1}^k}{k} \quad (4.44)$$

$$B = \begin{pmatrix} 1 & 1 & 0 \\ 1 & 1 & 0 \\ 0 & 0 & 0 \end{pmatrix} B \quad (4.45)$$

$$B = \sum_{n=1}^N \beta_n e_{31n}^* \frac{\alpha_n}{12} (h_n - h_{n-1})^3 \quad (4.46)$$

$$\gamma_k = (\gamma_k, \gamma_k, 0)^T \quad (4.47)$$

$$\gamma_k = \sum_{n=1}^N \beta_n e_{31n}^* (V_n - V_{n-1}) \left(\frac{h_n^k + h_{n-1}^k}{2} \right) \quad (4.48)$$

A^k are stiffness equivalent terms obtained by summing on the different layers. γ_k are piezoelectric loading terms. B are the coefficients which takes into account the effect of the variation of the electric field across the thickness on the strain, ensuring full electromechanical coupling. These equations are also valid for pure elastic layers under linear strain-stress relation provided that the piezoelectric coupling coefficient e_{ij} are set to 0.

4.3.1 Strong form

Equations (4.12) to (4.16), together with (4.39), allow to write the equations in local form, also called strong form. Combining linearly the x derivative of (4.15), the y derivative of (4.16) and (4.14), the shear forces Q_x and Q_y are discarded, which leads

to a condensed equation and a reduction of the unknowns:

$$M_{xx,xx} + 2M_{xy,xy} + M_{yy,yy} = p + I^1 \ddot{w} + I^2 (\ddot{u}_{0,x} + \ddot{v}_{0,y}) - I^3 (\ddot{w}_{xx} + \ddot{w}_{yy}) \quad (4.49)$$

Replacing the expressions for forces and moment in terms of displacement components in (4.12), (4.13) and (4.49), assuming constant equivalent stiffness coefficients and taking into account that $A_{11}^k = A_{12}^k + A_{66}^k$ in transversely isotropic material, three partial derivative equations are obtained for the three displacement component:

$$-(A_{11}^3 - B) \Delta \Delta w + A_{11}^2 \Delta (u_{0,x} + v_{0,y}) = p - \Delta \gamma_1 + I^1 \ddot{w} - I^2 (\ddot{u}_{0,x} + \ddot{v}_{0,y}) + I^3 \Delta \ddot{w} \quad (4.50)$$

$$-A_{11}^2 \Delta w_x + A_{11}^1 (u_{0,xx} + v_{0,yy}) + \frac{A_{66}^1}{2} (u_{0,yy} - v_{0,xx}) = -\gamma_{0,x} + I^1 \ddot{u}_0 + I^2 \ddot{w}_{,x} \quad (4.51)$$

$$-A_{11}^2 \Delta w_y + A_{11}^1 (u_{0,xy} + v_{0,yy}) + \frac{A_{66}^1}{2} (v_{0,xx} - u_{0,yy}) = -\gamma_{0,y} + I^1 \ddot{v}_0 + I^2 \ddot{w}_{,y} \quad (4.52)$$

$$(4.53)$$

These equations are quite complicated to handle due to the cross coupling. In order to simplify, let us first take the sum of the derivatives with respect to x of (4.51) and with respect to y of (4.52). A new relation is obtained:

$$-A_{11}^2 \Delta \Delta w + A_{11}^1 \Delta (u_{0,x} + v_{0,y}) = -\Delta \gamma_0 + I^1 (\ddot{u}_{0,x} + \ddot{v}_{0,y}) + I^2 \Delta \ddot{w} \quad (4.54)$$

which can be combined linearly with the equation obtained from the equilibrium of moments. Taking (4.50) and (4.54), multiplying them by A_{11}^1 and A_{11}^2 , respectively, and summing gives after some simplification:

$$I_E \Delta (\Delta w + \gamma_c) = p + I^1 \ddot{w} + I_c^2 (\ddot{u}_{0,x} + \ddot{v}_{0,y}) - I_c^3 \Delta \ddot{w} \quad (4.55)$$

where I_E is an equivalent bending stiffness, I_c^k are corrected inertia coefficients and γ_c is the equivalent piezoelectric load, given by:

$$I_E = \frac{(A_{11}^2)^2 - A_{11}^1 (A_{11}^3 - B)}{A_{11}^1} \quad (4.56)$$

$$I_c^k = \frac{I^k A_{11}^1 - I^{k-1} A_{11}^2}{A_{11}^1} \quad (4.57)$$

$$\gamma_c = A_{11}^1 \gamma_1 - A_{11}^2 \gamma_0 \quad (4.58)$$

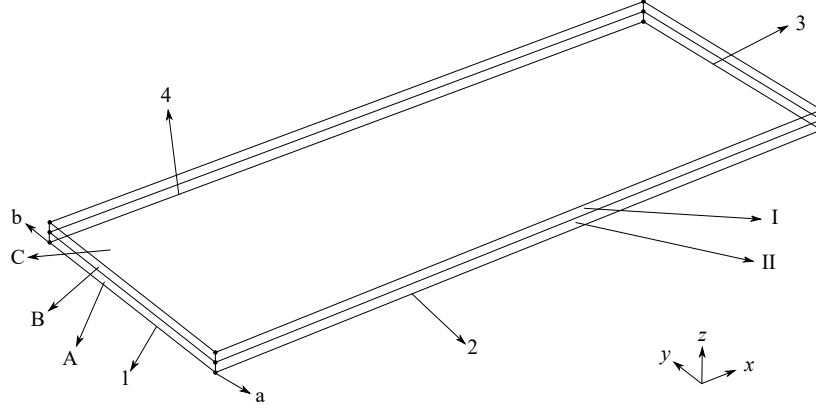


Fig. 4.6: Geometry of the case study. The domain I is made of PZT-5H and the domain II, of steel AISI 4340, for which physical parameters such as Young's modulus, piezoelectric coefficient, and so on are available in appendix A. Domain I is modelled using linear piezoelectricity and is polarized in the z direction, positively. For domain II, linear elasticity is used. At the interface plane B, displacement is supposed to be continuous and is electrically grounded. Plane B is put at a voltage of 200 V but is not mechanically loaded. The w displacement component is imposed to 0 at edges 1, 2, 3 and 4. In addition, u_0 and v_0 are also imposed to 0 at point a and u_0 is imposed to 0 at point b.

The main advantage of this formulation is that it decouples w from u_0 and v_0 in static case.

Static analysis of a simply supported bender

Eq. (4.55) has been used to study the static behaviour of a rectangular monomorph of 65 mm long, 25 mm wide and 1 mm thick, simply supported on its four sides and loaded by a constant voltage but no external pressure. A series solution is obtained for the problem using Fourier series expansion:

$$w(x, y) = \sum_{n=1}^{\infty} \sum_{m=1}^{\infty} \frac{-4\gamma_c}{I_E} \frac{\sin\left(\frac{n\pi x}{L}\right) \sin\left(\frac{m\pi y}{W}\right) ((-1)^{n-1} - 1) ((-1)^{m-1} - 1)}{nm\pi^2 \left(\left(\frac{n\pi}{L}\right)^2 + \left(\frac{m\pi}{W}\right)^2 \right)} \quad (4.59)$$

It is compared to a 3D FEM model developed in Comsol Multiphysics whose geometry is shown in fig. 4.6. The domain I is made of PZT-5H and the domain II, of

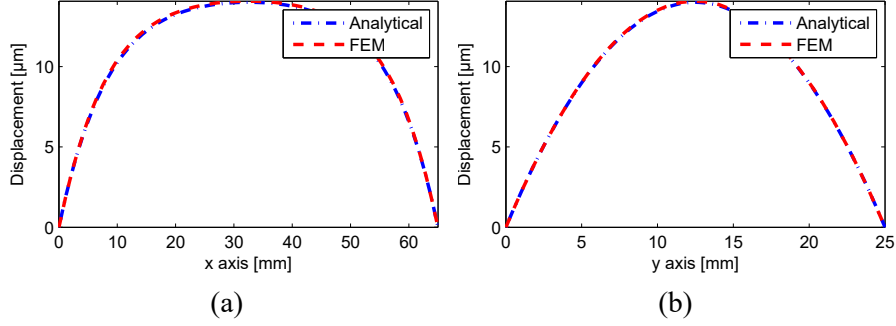


Fig. 4.7: Displacement of a simply supported rectangular plate under electrical load only (a) along x axis and (b) y axis on the median of the plate.

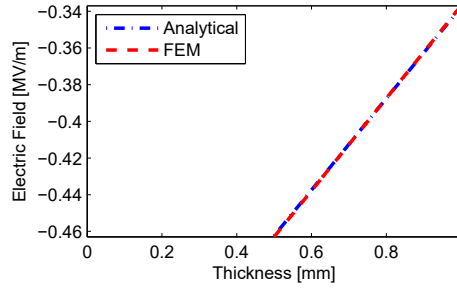


Fig. 4.8: Electric field across the plate thickness at the middle of the plate.

steel AISI 4340, for which physical parameters such as Young's modulus, piezoelectric coefficient, and so on are available in appendix A. Domain I is modelled using linear piezoelectricity and is polarized in the z direction, positively. For domain II, linear elasticity is used. At the interface plane B, displacement is supposed to be continuous and is electrically grounded. Plane B is put at a voltage of 200 V but is not mechanically loaded. The w displacement component is imposed to 0 at edges 1, 2, 3 and 4. In addition, u_0 and v_0 are also imposed to 0 at point a and u_0 is imposed to 0 at point b. Notes that the electrostatic physics is only used for the piezoelectric domain whereas the structural dynamics is computed for the whole geometry.

The mesh is automatically generated using built-in functionalities. 'General physics' setting is used with a predefined value of 'finer' to generate a tetrahedral mesh. The stationary study is chosen and the default direct solver, i.e. PARDISO solver, for fully coupled variables is used. A 'Cut line 3D' is used to plot the results along the middle axis in x and y , at $z = 0$, for the displacement w . An other is used to plot the results along the thickness at the middle of the plate.

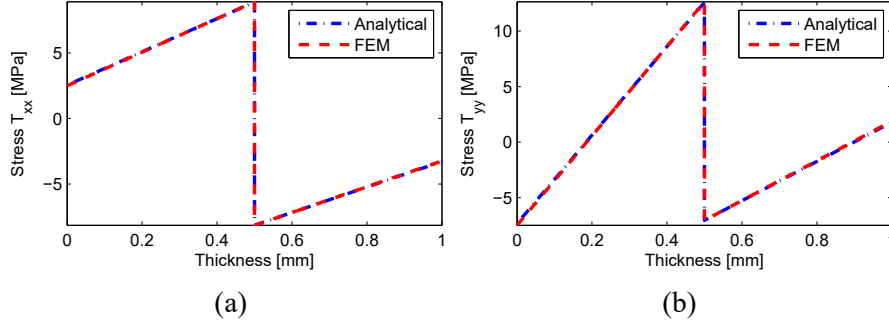


Fig. 4.9: Stress (a) T_{xx} and (b) T_{yy} across the layers at the middle of the plate.

Displacement along x and y axis on the median axis has been computed using the series solution too and are shown in fig. 4.7 (a) and (b). In addition, the electric field across the thickness has also been computed and is shown in fig. 4.8. And finally, the stresses T_{xx} and T_{yy} across the plate thickness at the middle of the plate are shown in fig. 4.9. A good agreement between the series solution and 3D model is observed in this case.

Unfortunately, using analytical methods to find a solution to more general problem is often intractable. Thus, the full model has not been used in its strong form in the micropump modelling mainly because it is not the best suited form to handle contact constraint as will be explained in the chapter 5.

4.3.2 Weak form

Equations in strong form are complicated to solve analytically in most of the cases. In order to solve them, many numerical methods can be used but here, a finite element method is preferred. It allows to solve the equations on more complicated domains, but also to take into account more complex situations, such as the ones where contact occurs, using generally known methods. In order to apply the finite element method, equations must be transformed to the weak form. This is done by applying the virtual work principle to our infinitesimal plate element. Equations (4.12) to (4.16) are multiplied by a virtual displacement $\tilde{u}$, $\tilde{v}$, $\tilde{w}$, $\tilde{w}_{,x}$ and $\tilde{w}_{,y}$ respectively and then integrated on the domain. However, some care has to be taken with sign conventions. Indeed, displacement w and rotations $w_{,x}$ and $w_{,y}$ are not independent. A displacement $\tilde{w}\hat{z}$ means rotations $-\tilde{w}_{,x}\hat{y}$ and $\tilde{w}_{,y}\hat{x}$ and must be respected in the virtual work formulation. An integration by part is performed giving:

$$\begin{aligned}
& - \int_{\Omega} N_{xx} \tilde{u}_{,x} + N_{yx} \tilde{u}_{,y} + (I^1 \ddot{u}_0 - I^2 \ddot{w}_{,x}) \tilde{u} d\Omega \\
& + \int_{\Gamma_N} \tilde{u} \vec{t}_x \cdot \vec{n} d\Gamma = 0
\end{aligned} \tag{4.60}$$

$$\begin{aligned}
& - \int_{\Omega} N_{xy} \tilde{v}_{,x} + N_{yy} \tilde{v}_{,y} + (I^1 \ddot{v}_0 - I^2 \ddot{w}_{,y}) \tilde{v} d\Omega \\
& + \int_{\Gamma_N} \tilde{v} \vec{t}_y \cdot \vec{n} d\Gamma = 0
\end{aligned} \tag{4.61}$$

$$\begin{aligned}
& - \int_{\Omega} Q_x \tilde{w}_{,x} + Q_y \tilde{w}_{,y} + p \tilde{w} + I^1 \ddot{w} \tilde{w} d\Omega \\
& + \int_{\Gamma_N} \tilde{w} \vec{q} \cdot \vec{n} d\Gamma = 0
\end{aligned} \tag{4.62}$$

$$\begin{aligned}
& \int_{\Omega} M_{yx} \tilde{w}_{,xy} + M_{xx} \tilde{w}_{,xx} + (I^2 \ddot{u}_0 - I^3 \ddot{w}_{,x}) \tilde{w}_{,x} \\
& + Q_x \tilde{w}_{,x} d\Omega - \int_{\Gamma_N} \tilde{w}_{,x} \vec{m}_x \cdot \vec{n} d\Gamma = 0
\end{aligned} \tag{4.63}$$

$$\begin{aligned}
& \int_{\Omega} M_{yx} \tilde{w}_{,xy} + M_{yy} \tilde{w}_{,yy} + (I^2 \ddot{u}_0 - I^3 \ddot{w}_{,y}) \tilde{w}_{,y} \\
& + Q_y \tilde{w}_{,y} d\Omega - \int_{\Gamma_N} \tilde{w}_{,y} \vec{m}_y \cdot \vec{n} d\Gamma = 0
\end{aligned} \tag{4.64}$$

where $\vec{t}_x$, $\vec{t}_y$, $\vec{q}$, $\vec{m}_x$ and $\vec{m}_y$ are known loading terms, imposed on a part of the boundary. The first two are thus traction forces, the third is a shear loading and the last two are applied torques. These loading terms are, in component form:

$$\vec{t}_x = (N_{xx}, N_{xy}) \tag{4.65}$$

$$\vec{t}_y = (N_{xy}, N_{yy}) \tag{4.66}$$

$$\vec{m}_x = (M_{xx}, M_{xy}) \tag{4.67}$$

$$\vec{m}_y = (M_{xy}, M_{yy}) \tag{4.68}$$

$$\vec{q} = (M_{xx,x} + M_{xy,y}, M_{xy,x} + M_{yy,y}). \tag{4.69}$$

$$\tag{4.70}$$

On the other part of the boundary, Γ_D , the displacement is imposed for u_0 and v_0 . For the last component of the displacement, the displacement itself, w , can be fixed, but also its gradient. Of course, mixed conditions are possible where the displacement is fixed and the applied torques such as for simply supported conditions.

Looking more closely to (4.62) to (4.64), it appears they can be condensed to

eliminate the internal shear forces Q_x and Q_y . It gives the single equation:

$$\begin{aligned} \int_{\Omega} & M_{xx} \tilde{w}_{,xx} + 2M_{yx} \tilde{w}_{,xy} + M_{yy} \tilde{w}_{,yy} - p \tilde{w} \\ & - I^1 \ddot{w} \tilde{w} + I^2 (\ddot{u}_0 \tilde{w}_{,x} + \ddot{v}_0 \tilde{w}_{,y}) \\ & - I^3 (\ddot{w}_{,x} \tilde{w}_{,x} + \ddot{w}_{,y} \tilde{w}_{,y}) d\Omega \\ & - \int_{\Gamma_N} (\tilde{w}_{,x} \vec{m}_x + \tilde{w}_{,y} \vec{m}_y - \tilde{w} \vec{q}) \cdot \vec{n} d\Gamma = 0 \end{aligned} \quad (4.71)$$

So (4.60), (4.61) and (4.71) are the final forms of the weak formulation for the dynamics of the plate.

4.4 Discretization

Either the strong or the weak form can be used and discretized to find a solution but, it has an implication, mathematically, since the weak form relaxed the condition on differentiability of the solution. This means a solution may not exist for the strong form, but the same problem can be solved in the weak form. This has a special importance when dealing with contact and will be explained in chapter 5. That is why the weak form has been chosen for the development of the numerical scheme. Remarks (4.60), (4.61) and (4.71) can be used in a software, such as Comsol Multiphysics, but here, it has been chosen to develop the code in Matlab. The equations are discretized using conforming finite element for space and a generalized alpha method for time integration. This will lead to a linear system of algebraic equations for incremental displacement.

4.4.1 Spatial discretization

In finite element, the domain is partitioned in simple geometrical subdomains, such as triangle or rectangle in 2D called mesh. For the sake of simplicity, let's consider, without any additional details, that u_0 , v_0 and w are approximated by polynomials, which are nonzero only in the subdomain where they are defined, and have some continuity requirements from their domain of definition and with their neighbouring elements. More explanations on the element used and some computational aspects are developed in appendix B. The approximations can be written as:

$$u_h(\xi, \eta) = \Phi^T \mathbf{U} \quad (4.72)$$

$$v_h(\xi, \eta) = \Phi^T \mathbf{V} \quad (4.73)$$

$$w_h(\xi, \eta) = \Psi^T \mathbf{W} \quad (4.74)$$

where Φ , Ψ are the polynomials used to interpolate the nodal values given in vectors $\mathbf{U}$ for discrete values of u_0 , $\mathbf{V}$ for discrete values of v_0 and $\mathbf{W}$ for discrete values of w , $w_{,x}$, $w_{,y}$ and $w_{,xy}$. These approximations can be used to compute the integrals of

(4.60), (4.61) and (4.71) and are used to represent the approximation of the solution, but also the virtual displacement. After assembling all these equations this leads to:

$$\tilde{\mathbf{d}} \left(\mathbf{M}\ddot{\mathbf{d}} + \mathbf{K}\mathbf{d} + \mathbf{F} \right) = 0 \quad (4.75)$$

which should be verified for all admissible virtual displacement $\tilde{\mathbf{d}}$. $\mathbf{d}$ is the vector containing the nodal values for all degrees of freedoms. $\mathbf{M}$ is generally called the mass matrix, $\mathbf{K}$ is the stiffness matrix and $\mathbf{F}$ is the loading vector and have the following structure:

$$\tilde{\mathbf{d}} = \begin{bmatrix} \tilde{\mathbf{U}}_0 & \tilde{\mathbf{V}}_0 & \tilde{\mathbf{W}} \end{bmatrix} \quad (4.76)$$

$$\mathbf{M} = \begin{pmatrix} -\mathbf{M}_{uu} & 0 & \mathbf{M}_{uw} \\ 0 & -\mathbf{M}_{vv} & \mathbf{M}_{vw} \\ \mathbf{M}_{uw}^T & \mathbf{M}_{vw}^T & -\mathbf{M}_{ww} \end{pmatrix} \quad (4.77)$$

$$\mathbf{K} = \begin{pmatrix} -\mathbf{K}_{uu} & -\mathbf{K}_{uv} & \mathbf{K}_{uw} \\ -\mathbf{K}_{uv}^T & -\mathbf{K}_{vv} & \mathbf{K}_{vw} \\ \mathbf{K}_{uw}^T & \mathbf{K}_{vw}^T & -\mathbf{K}_{ww} \end{pmatrix} \quad (4.78)$$

$$\mathbf{F} = \begin{pmatrix} \mathbf{F}_u \\ \mathbf{F}_v \\ \mathbf{F}_w \end{pmatrix} \quad (4.79)$$

Each submatrix shows the coupling between the different displacement components and is detailed in appendix B. In fine a system of ordinary differential equations is obtained:

$$\mathbf{M}\ddot{\mathbf{d}} + \mathbf{K}\mathbf{d} + \mathbf{F} = 0. \quad (4.80)$$

4.4.2 Time discretization

The semidiscrete system (4.80) has to be integrated in time. A generalized alpha method, as in Hartmann et al. (2007), is used. $\mathbf{d}$, $\dot{\mathbf{d}}$ and $\ddot{\mathbf{d}}$ are chosen as state variables and, using a Newmark scheme, are rewritten to express $\dot{\mathbf{d}}_{n+1}$ and $\ddot{\mathbf{d}}_{n+1}$ in terms of $\delta\mathbf{d}_n = \mathbf{d}_{n+1} - \mathbf{d}_n$, $\dot{\mathbf{d}}_n$ and $\ddot{\mathbf{d}}_n$:

$$\dot{\mathbf{d}}_{n+1} = \frac{\gamma}{\beta\delta t}\delta\mathbf{d}_n - \left(\frac{\gamma}{\beta} - 1\right)\dot{\mathbf{d}}_n - \left(\frac{\gamma}{2\beta} - 1\right)\delta t\ddot{\mathbf{d}}_n \quad (4.81)$$

$$\ddot{\mathbf{d}}_{n+1} = \frac{1}{\beta\delta t^2}\delta\mathbf{d}_n - \frac{1}{\beta\delta t}\dot{\mathbf{d}}_n - \left(\frac{1}{2\beta} - 1\right)\ddot{\mathbf{d}}_n \quad (4.82)$$

A generalized mid-point discretization for (4.80):

$$\mathbf{M}\ddot{\mathbf{d}}_{n+1-\alpha_m} + \mathbf{K}\mathbf{d}_{n+1-\alpha_f} + \mathbf{F}(\dot{\mathbf{d}}_{n+1-\alpha_f}) = 0 \quad (4.83)$$

with:

$$\ddot{\mathbf{d}}_{n+1-\alpha_m} = (1 - \alpha_m) \ddot{\mathbf{d}}_{n+1} + \alpha_m \ddot{\mathbf{d}}_n \quad (4.84)$$

$$\mathbf{d}_{n+1-\alpha_f} = (1 - \alpha_f) \mathbf{d}_{n+1} + \alpha_f \mathbf{d}_n \quad (4.85)$$

where δt is the time step and the parameters are defined according to another parameter, $0 \leq \rho_\infty \leq 1$, which fixes the numerical dissipation of high frequency terms, is used:

$$\alpha_m = \frac{2\rho_\infty - 1}{\rho_\infty + 1} \quad (4.86)$$

$$\alpha_f = \frac{\rho_\infty}{\rho_\infty + 1} \quad (4.87)$$

$$\beta = \frac{1}{4}(1 - \alpha_m + \alpha_f)^2 \quad (4.88)$$

$$\gamma = \frac{1}{2} - \alpha_m + \alpha_f \quad (4.89)$$

A value of one for ρ_∞ ensures energy conservation. The load vector is dependent of the displacement rate through the current equation (4.36), for the laminated plate containing piezoelectric material. For the laminated plate under Lorentz force, the load vector is dependent of the displacement through the variation of the magnetic field with the position of the plate. It is now possible to find a system of equations in incremental displacement $\delta \mathbf{d}$:

$$\mathbf{K}_{\text{eff}} \delta \mathbf{d}_n = \mathbf{g}_n \quad (4.90)$$

with:

$$\mathbf{K}_{\text{eff}} = \frac{1 - \alpha_m}{\beta \delta t^2} \mathbf{M} + (1 - \alpha_f) \mathbf{K} \quad (4.91)$$

$$\begin{aligned} \mathbf{g}_n = & (1 - \alpha_f) \mathbf{F}_{n+1} + \alpha_f \mathbf{F}_n - \mathbf{K} \mathbf{d}_n \\ & - \mathbf{M} \left(\frac{\alpha_m - 1}{\beta \delta t} \dot{\mathbf{d}}_n + \left(1 - \frac{1 - \alpha_m}{2\beta} \right) \ddot{\mathbf{d}}_n \right) \end{aligned} \quad (4.92)$$

Equation (4.90) together with (4.81) and (4.82) allows to compute all quantities of interest at nodal points.

4.5 Numerical study

The proposed numerical scheme has been used to analyse the behaviour of a piezoelectric monomorph subjected to transient loads in two situations. Firstly as an actuator and secondly as an energy harvester, delivering power to an electrical load. The geometry of the model is composed of a first rectangular block, the domain I

as shown in fig. 4.10, of 60mm long, 25mm wide and 1mm thick. On the top of it, a second rectangular block is fixed, the domain II, of 55mm long, 25mm wide and 0.5mm thick is fixed. The first domain is modelled using linear elasticity with AISI 4340 steel and the second one use linear piezoelectricity with PZT-5H. The bonding between the two domains is such that displacement is continuous. Electrodes are not modelled, and thus, nor meshed, but instead a boundary condition imposing a voltage is used. On the interface between the two blocks, the surface is electrically grounded. For the upper electrode, called face A, a voltage is imposed whose value depend of the case study. A pressure load is imposed on faces A and B, whose value depends of the case study. Displacement of face C is fully constrained and imposed to 0. The previously introduced numerical scheme is implemented in Matlab. Since the developed equations consider constant coefficients, the domain is decomposed in two: one with only the elastic layer and a second with the two layers. Continuity of displacement and of slope for w is imposed at the interface. The mesh is composed of 210 rectangular elements.

All situations have been compared to a 3D simulation using COMSOL Multiphysics, where mechanical variables, and thus structural dynamic physics, are solved for the whole geometry, the electrical equations from electrostatics are solved only for the piezoelectric domain. The mesh and solvers used are study dependent and will be explained in each case.

4.5.1 Actuator

As the two first applications, the monomorph is loaded by a pressure which varies sinusoidally at 100Hz and 750Hz with a peak value of 3000Pa. In addition a voltage of 100V which varies sinusoidally at the same frequencies is also applied.

A mesh made of tetrahedral elements with predefined size of ‘coarse’ is used for this analysis. The study is a transient one made on 3 periods with zero initial condition for displacement and speed. The solver used is the one proposed by default, i.e. the multifrontal massively parallel sparse direct solver (MUMPS) and all the variables are solved as fully-coupled.

The displacement components are shown in fig. 4.11a, 4.11b, 4.11c, 4.12a, 4.12b and 4.12c. They are observed at the lower right corner of the monomorph, marked as ‘a’ on fig. 4.10. As can be seen, the results are in quite good agreement. At 750 Hz, a higher frequency is clearly visible on the v_0 component. This high frequency oscillation is the expression of one of the eigen mode. They appear in all transient study of such a system as a part of the response which is normally damped. In this case, only numerical damping, which is introduced through the generalized alpha method, is present and its effect can be noticed in longer simulation time. A spectral analysis performed on both model for the three displacement components shows that the

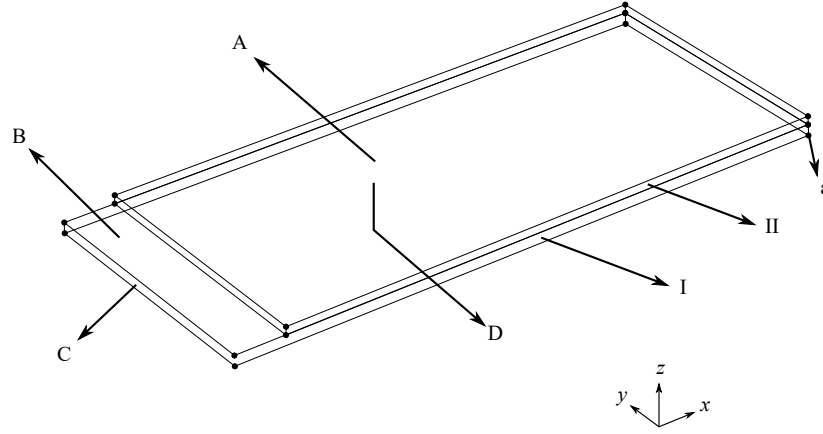


Fig. 4.10: Geometry of the case study. Composed of a first rectangular block, the domain I, of 60mm long, 25mm wide and 1mm thick. On the top of it, a second rectangular block is fixed, the domain II, of 55mm long, 25mm wide and 0.5mm thick is fixed. The first domain is modelled using linear elasticity with AISI 4340 steel and the second one use linear piezoelectricity with PZT-5H. Electrodes are not modelled, but instead a boundary condition imposing a voltage is used. On the interface between the two blocks, the surface is electrically grounded. For the upper electrode, called face A, a voltage is imposed whose value depend of the case study. A pressure load is imposed on faces A and B, whose value depends of the case study. Displacement of face C is fully constrained and imposed to 0.

response is composed of a ray at 750 Hz but also a series of other rays at 238 Hz and 1545 Hz for u_0 , 238 Hz, 1544 Hz and 9911 Hz for v_0 and 238 Hz and 1554 Hz for w . These are seen in response computed by the 2D and 3D models. Some additional rays appear in the 3D model which are certainly due to numerical noise. They do not correspond to eigen mode computed by Comsol, using an eigen frequency study.

The model has been implemented using Matlab software. The computation is achieved in less than 6 seconds. In comparison the model using Comsol has taken more than 200 seconds. Some snapshot of the displacement w are also shown at 750Hz in fig. 4.14a, 4.14b, 4.14c, 4.14d, 4.14e and 4.14f.

4.5.2 Energy harvester

In the case of energy harvesting the applied voltage is unknown and depends of the connected electrical load. For this study a 12 k Ω resistance is connected to the ben-

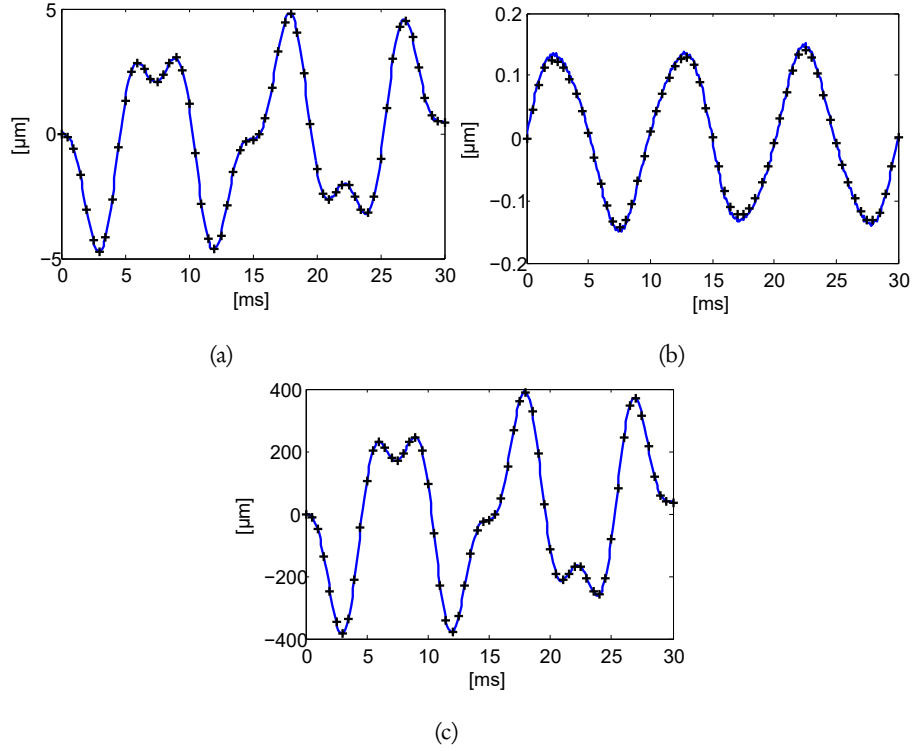


Fig. 4.11: (a) u_0 , (b) v_0 and (c) w displacement components at 100 Hz. The blue line is the current method. The black cross are the 3D Comsol study.

der. The external excitation used is an applied pressure of 300Pa which varies sinusoidally at 750Hz. Eq (4.36) is integrated using built in Matlab solvers for ode to obtain the applied voltage at time step $n + 1$, based on the speed computed at previous time step n . So the electrical equation and the mechanical ones are only loosely coupled, since there is no subiteration to ensure the strict convergence between the electrical and mechanical equations.

In Comsol, an additional physics interface, called ‘electrical circuit’, has to be used to take into account the resistive load. Instead of imposing a voltage condition on the upper electrode, the face A, in the ‘electrostatics’ interface, a condition called ‘terminal’ is imposed. This one computes the integral of the electric displacement on the face and its time derivative to obtain the current, which can be used in the ‘electrical circuit’ interface. A ‘resistor’ condition is added in the interface with the 12 k Ω value. The mesh used for this study is a tetrahedral one with the predefined size value ‘normal’. The physics are solved in a segregated way, solving in a first step the current through the resistive load and in a second step the displacement and voltage generated by the bender. Each system is solved using the MUMPS solver.

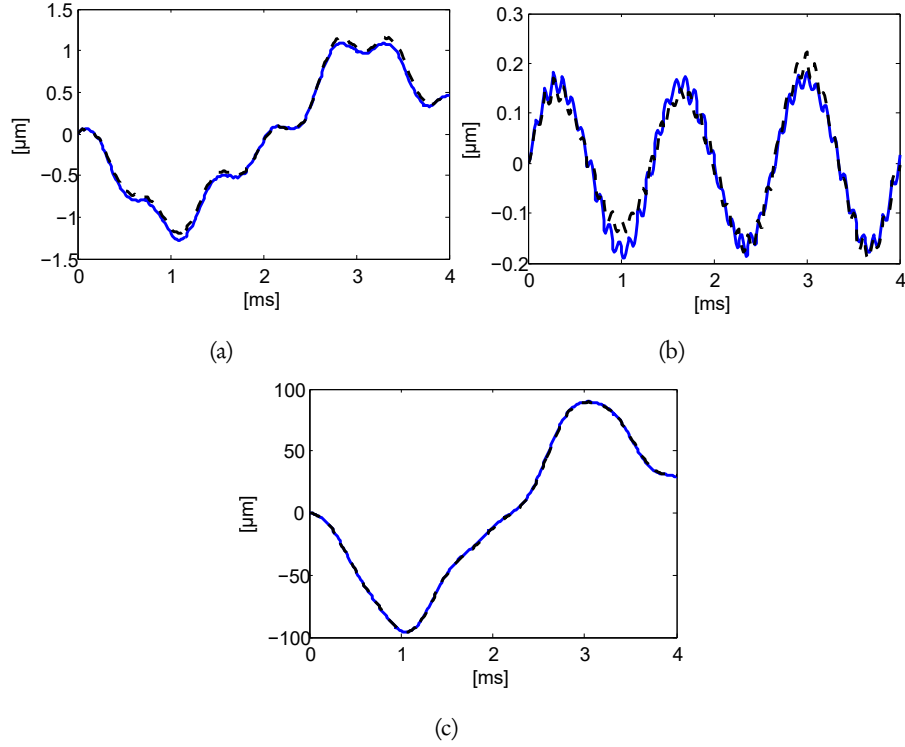


Fig. 4.12: (a) u_0 , (b) v_0 and (c) w displacement components at 750 Hz. The blue line is the current method. The black dashed line is the 3D Comsol analysis.

This is the default strategy proposed by Comsol.

The obtained voltage and its comparison with the one obtained from Comsol is shown in fig. 4.15. Once again the simulations are in quite good agreement. The computation has taken less than 10 s using Matlab software but the 3D model has taken more the 1200 s showing the huge saving in computational time.

4.5.3 Study of stresses and electric field

Stresses and electric field are computed at steady state, under constant voltage load and without any external pressure. von Mises stress are computed using :

$$\sigma = \sqrt{(T_{xx}^2 + T_{yy}^2 - T_{xx}T_{yy} + 3T_{xy}^2)}. \quad (4.93)$$

It is compared to 3D Comsol model, the same as used for the actuator study but with a constant pressure and voltage instead of a time varying one. Computing L^2 -norm

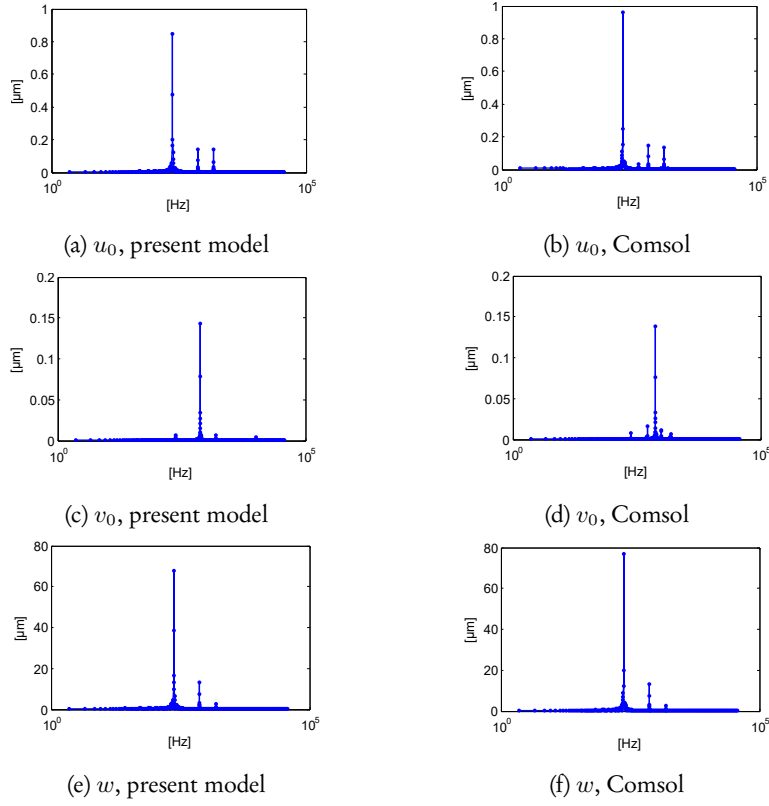


Fig. 4.13: Spectral analysis made on the three displacement components, for the present model and the 3D model in Comsol.

error on von Mises stress and electric field gives respectively 0.1 and 0.042 percent for piezoelectric material. For the elastic material it is 0.34 percent.

4.6 Conclusion

In this chapter, the constitutive equations for coupled bending-extension of thin laminated plates under bending loads due to external pressure and piezoelectric effects have been derived, for actuating but also sensing purposes. The electrical equations are solved analytically and used in the mechanical equations. This reduces the number of unknowns to compute numerically. The mathematical developments to find the electric field expression has shown the link between the expansion used with respect to the thickness coordinate for the displacement and the one which should be used for the electric potential, which was not clear in the literature. It is believed that the method can be applied to more evolved theories presented in the literature, reducing the number of unknowns to solve. In addition, compared to other simplified

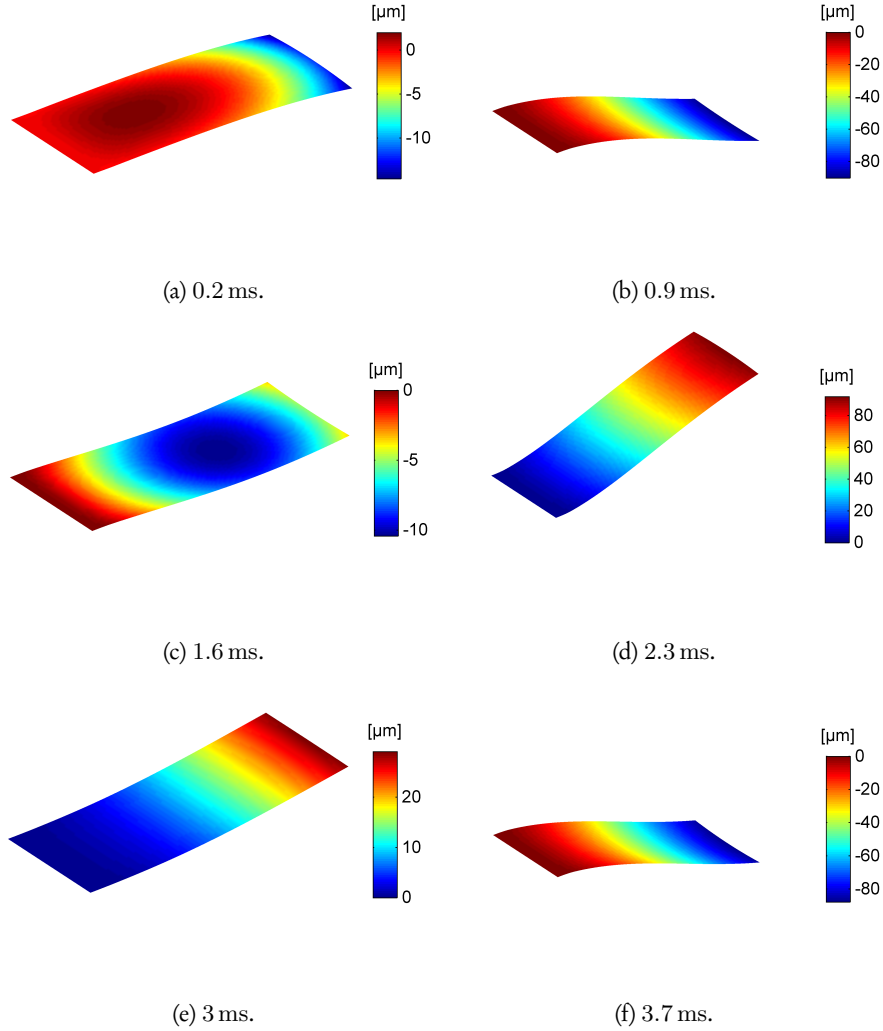


Fig. 4.14: w displacement component at 750 Hz at various simulation time.

models where the electric field is assumed constant across the piezoelectric layer, the developed equations for the electric field allow to compute a far more accurate value for the stresses as already noted in the first part and as shown here in the numerical results. Compared to the models developed in the first part, the dynamics of the plate is now taken into account and stresses compare well with the ones of the 3D model. These will be studied deeper in the chapter 6, where the developed theory is applied to the micropump concepts proposed as benchmark cases.

Based on a weak form of these equations, a finite element discretization in space

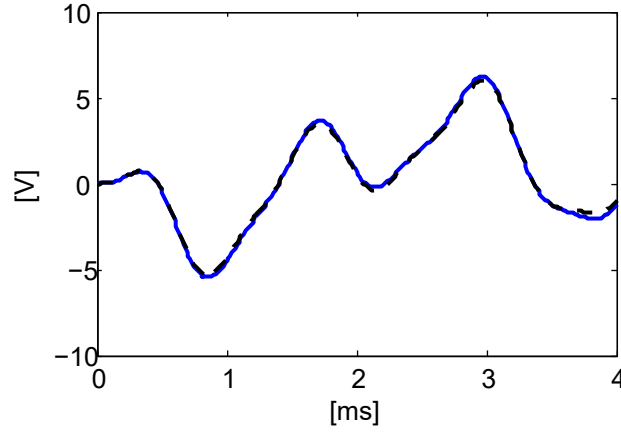


Fig. 4.15: Voltage generated at 750Hz. The blue line is the current method. The black dashed line is the 3D Comsol study.

and a generalized alpha method in time has been proposed for integrating the mechanical equations. The electrical equation is solved only if the voltage is not directly imposed. It is not solved using a monolithic approach. Indeed the stiffness of the electrical equation imposes a too high constraint on the time step. Thus, instead, the speed at the end of the mechanical time step n is used to compute the voltage at time $n + 1$ using the electrical equation (4.36). This voltage will be used to compute the speed. A built in ode solver from Matlab has been used here. The proposed solver is thus a partitioned, loosely coupled one.

The finite element discretization and some details for the implementation have been given in appendix B. The proposed method has been compared to the a 3D model implemented in Comsol Multiphysics and shows a good agreement, especially for stresses computation which were not well physically modelled by one dimensional approach. In addition, the proposed method can lead to a huge saving in computational time, as seen for the harvester application where it takes less than 10 s using Matlab and more than 1200 s for the full 3D model.

Treatment of frictionless contacts

5

The term ‘contact’ covers different situations: with friction or not, with adhesion or not. Here, only frictionless contact with a rigid obstacle and no adhesion is considered. It means the body coming into contact with the plate will only exert a normal force on it to avoid penetration, but will not retain the plate to move away from the obstacle, neither to slip, since all tangential stresses are null. In addition, only contact is dealt with, which means that, even if dynamics is used for plates’ equations, the developed method does not consider impact and energy conservation, or dissipation, occurring in these cases.

Handling such constraints is often quite challenging since, from a mathematical point of view, it is a nonsmooth problem which requires special algorithms to be solved. In the literature, the problem of a plate in frictionless contact with another body is also known as a kind of obstacle problem. If the obstacle problem is deeply treated for a membrane i.e. a structure modelled by a laplacian, it is not the case for bending plates, which are, in most simple cases, modelled by a bilaplacian. Indeed, the problem is much more complicated from a mathematical point of view, as reported in Pozzolini (2009). Another way of seeing this problem is to consider the plate displacement with an unknown boundary where three boundary conditions are imposed. Solving the plate problem only requires two boundary conditions but, since the boundary shape and position is also part of the problem, a third information is needed to find it. This is the kind of approach used for treating the contact with beams, in the first part of this thesis. This kind of problem is called the free boundary problem. A third term which is encountered is mixed linear complementarity problem, which comes from the fact that the plate can deflect normally or be in contact but not both at the same time. Notes that the term ‘mixed linear complementarity problem’ covers many other problems than this one. A fourth term used for this kind of problem is the Signorini problem.

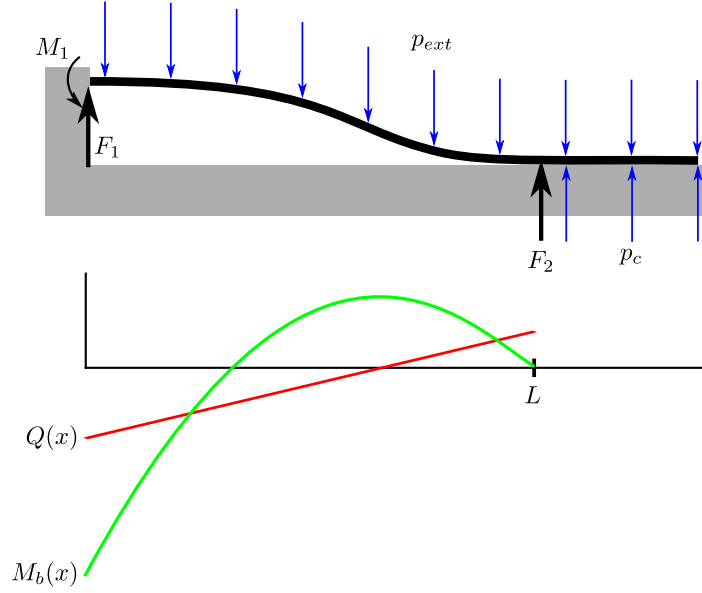


Fig. 5.1: Beam in contact with a flat obstacle with constant line load and its internal effort. The beam conforms to the obstacle on the contact zone and the contact pressure compensates exactly the external pressure applied on top of the beam. A punctual force F_2 appears at the interface, ensuring equilibrium. Since the contact is frictionless, the obstacle can not applied a bending moment, meaning the internal moment is continuous at the interface.

This chapter first explains what plate contact means and which kind of situation can arise. Then an overview of the different formulation and algorithms presented in the literature is given. The third section explains how the contact constraint is handled, and extended to the study of the dynamics of the plate, for modelling the micropumps.

5.1 Geometrical and equilibrium aspects

Let us first consider the case treated in chapter 3 of a beam in contact with a rigid and flat obstacle as seen in fig. 5.1. This is sufficient to understand the physics of the problem, extending it to equations of plates being rather a problem of solving the equations in a 2D general geometry. In the case of beams, it was solved analytically by considering a beam of length L , with zero displacement, slope and curvature or bending moment at both ends. In fact, only two boundary conditions can be prescribed at each end, or the problem will be over constrained. As was said, there is no problem at all since L is also part of the problem and is computed considering the third boundary condition has to be fulfilled.

Behind these boundary conditions, an hypothesis is implicitly made: the contact is conformal. It means the deflection of the beam follows exactly the obstacle on the contact zone. If the contact occurs where piezoelectric patch are not powered, it means the remaining part of the beam is simply at rest and a counter pressure compensates exactly the pressure applied on top of the plate. In addition, a shear force will appear, at the interface with the contact zone, to ensure the equilibrium of the beam.

However, if the contact occurs in a zone where the piezoelectric patch is submitted to a non-zero voltage, the constant curvature of the piezoelectric effect has to be compensated exactly, which means a constant bending moment has to be applied externally. This will not occur if the applied pressure is constant or evolves linearly. In this case, the contact can not be conformal and the null curvature or bending moment can not be applied at the contact point. Thus, it seems this boundary condition is not the best suited to represent the contact behaviour in a general case.

Thinking of the bending of the beam in a general case, four boundary conditions can be applied: fixing the displacement, the normal slope, the bending moment or the shear force. In case of contact with an obstacle, the displacement at the contact point is automatically known. In addition, due to its bending stiffness, it is known that the jump in slope is null. For bending moment, the obstacle can not fix the condition since, in order to oppose a bending moment, the obstacle should be able to apply tangential load on the plate. As the contact is supposed to be frictionless, this is impossible. Thus, the only possible condition on the bending moment is a zero jump in its value across the contact boundary. For the shear force, an additional condition has to be fulfilled, as for the counter pressure in case of a conformal contact, it has to be positive i.e. to repel the plate.

In the case of a plate under obstacle constraint, the problem is similar but is much more complicated to deal with due to its 2D nature. The geometry of the contact set can be of three types this time: on one point, along a curve or a surface. Solving obstacle problem is rather difficult due to the low regularity of the solution. As stated in Pozzolini (2009), the solution for bilaplacian obstacle problem is only C^2 and not C^4 , as required by the local form. This explains why, in the literature, the weak form of the equation is preferred for obstacle problem dealing with plate. However, if the solution is not C^4 on the whole domain, it is piecewisely C^4 on the subdomains defined by the contact interface, meaning that if the interface is known, a solution using the strong form can be computed.

The contact conditions can be stated as follows:

$$w - \Psi \geq 0 \quad (5.1)$$

$$p_c \geq 0 \quad (5.2)$$

$$(w - \Psi) p_c = 0 \quad (5.3)$$

where Ψ is the obstacle position and p_c is the counter pressure in case of conformal contact and are thus conditions for the plate domain. Physically, the first equation states that the plate has to be above the obstacle. Note that treating the case where the plate is under the obstacle is straightforward. The second equation means the pressure imposed by the obstacle on the plate, to prevent interpenetration of them, can not retain the plate since no adhesion is taken into account. The last equation, called the complementarity condition, states that the rigid obstacle will exert a pressure only if there is a contact. Additional conditions have to be fulfilled with respect to the contact interface:

$$w - \Psi = 0 \quad (5.4)$$

$$\llbracket Q_n \rrbracket \geq 0 \quad (5.5)$$

$$\llbracket M_n \rrbracket = 0 \quad (5.6)$$

$$\llbracket \nabla w \rrbracket = 0 \quad (5.7)$$

where $\llbracket \cdot \rrbracket$ denotes the jump in the quantity $\cdot$, across the interface.

Solving the plate contact problem analytically seems intractable. First because, based on the strong form of the equation, the problem has to be solved on a general domain whose topology is uncertain. If the topology can be guessed in some situations, such as in the static case of the micropumps, where, for the piezoelectric one, the contact line does not cut any actuated piezoelectric patch, the problem has still to be solved on a domain which will not be a simple geometric shape and thus where it is difficult to find a series or closed form solution. In addition, since the domain is unknown, a parametrization of the boundary will have to be introduced, which is an additional difficulty.

5.2 Overview of literature

A deep research has been made in the literature to find a proper method to solve the described problem which is known under various names. These ones sometimes covered much more general problem than simply the case of a thin plate coming into contact with another body, rigid or not. Instead of giving a true state of the art about the problem, it is preferred to give here a glance at the various formulations of the problem and to some techniques, discretization and algorithms that could be used to solve it. For each method, some references to the literature will be given and

if possible to those dealing with plates. Even if an effort has been made to be the most exhaustive possible, the subject is too large to ensure that all is covered here. The research in the literature has been lead considering the different names associated to the problem i.e. obstacle problem, contact mechanics, signorini problem, mixed linear complementarity problem and free boundary problem.

In this section, a first point is dedicated to classify the various approaches encountered in the literature, following the formalism used to treat the problem. Then, the methods used to discretize the problem are reviewed.

5.2.1 Formalisms and methods

According to the literature, it seems that all formalisms are linked in one way or another to a constrained optimization problem, for which various techniques have been developed. These approaches can be separated in two groups, depending on the functional to minimize and of the type of constraint.

Elastic energy functional

The first, and probably the most natural one, is to minimize the elastic energy of the plate under inequality constraint on the displacement. Once the problem is discretized, usually thanks to the finite element method, the optimization is equivalent to a quadratic programming problem, for which several tools are available.

The constrained problem can be transformed to a series of unconstrained ones through a penalty method, such as in Scholz (1984), where it is applied to the membrane case. Physically, it corresponds to introduce a spring like force, which is proportional to the penalty parameter k_s , where a contact occurs. A force is thus applied only if there is a penetration of the plate in the obstacle. Thus, with this method, the constraint is more respected if the parameter is bigger but, it will never be perfectly respected. In addition, a too large value destabilize the method. This has the advantage to be simple and does not introduce any additional unknowns.

Another kind of penalty method, which is also known under the names ‘barrier method’, or ‘interior penalty method’ or simply ‘interior point method’, has also been used to solve plate obstacle problems, such as in Fernandes et al. (2001) or Brenner et al. (2012). In this article, the problem has been discretized using finite elements. The problem is then reformulated as a mixed linear complementarity problem. The algorithm seeks a solution by following a path μ_k , which is linked to the complementarity condition. At each iteration, the inequality conditions are maintained and μ_k is decreased. As this parameter tends to zero, the algorithm approaches the solution of the problem. The algorithm use two quantities to evaluate its convergence, the norm of the residual of the discretized equations of the plate

loaded by external loads and by the counter pressure and the sum of path parameter μ_k at the current iteration.

Another method is to introduce Lagrange multiplier, which will represent the counter pressure exerted by the obstacle on the plate. Karush-Kuhn-Tucker conditions have to be fulfilled to be a solution of the problem. These conditions have a physical meaning and have already been presented since it corresponds to (5.1) to (5.3), with in addition, the equations of the plate with an additional pressure load, the Lagrange multiplier. The multipliers are additional unknowns but the contact conditions will be exactly fulfilled. However, using Lagrange multipliers does not give an explicit algorithm, but rather a system of both equalities and inequalities to solve. For this system, several approaches have been found in the literature. A first one is to transform them to a system of nonlinear and nonsmooth equations through the use of a nonlinear complementarity function (NCP-function). These functions have the particularity to cancel only if the inequalities, as well as the complementarity condition, are fulfilled. An example of such a function is:

$$\phi(a, b) := \min(a, b) = 0 \iff a \geq 0, b \geq 0 \text{ and } ab = 0. \quad (5.8)$$

These equations are nonsmooth, so classical Newton method fails but, some modified version of it have been developed to solve the problem, such as in Chen et al. (2000). An other method developed by Liu and Atluri (2008) is to transform the nonlinear system to a system of ODEs using a fictitious time and has the advantage that it does not require to solve a linear system of equation at each step. An other way to solve the system of nonlinear equations is based on homotopy theory. It is used to solve obstacle problems in the membrane case in Fan et al. (2010). In this paper a scalar homotopy map $h(\mathbf{x}, t)$ based on a parameter t is used. A system of ODEs is obtained from the map and then solved using an explicit Euler method. The idea is that starting from $t = 0$, where $h(\mathbf{x}_0, 0) = 0$, and then the map is followed, keeping $h(\mathbf{x}, t)$ to 0, until $t = 1$ where the solution to the nonlinear system of equation is obtained. Another method to solve the system of inequalities is to separate the points in two sets. The first one, called active set, regroups the points where the plate is in contact with the obstacle. The displacement is fixed and an approximation for the contact reaction, i.e. the Lagrange multiplier, is computed. The second one is the inactive set on which an approximation for the displacement is computed considering a null contact reaction. If a point violates an inequality, it moves from its set to the other. It iterates until both sets satisfies the inequalities. Since each point is in one and only one set, the complementarity condition is fulfilled at each iteration. Since in each sets the value of either the displacement or the lagrange multiplier are known, the number of unknowns is not changed compare to the unconstrained problem. This method has been used for contact problem arising with thin-walled structures in Hartmann et al. (2007). A variation of this type of algorithm is the block principal pivoting algorithm, as presented in Fernandes et al.

(2001), where the number of points allowed to move from one set to the other is controlled dynamically to reduce the number of iterations.

The augmented lagrangian method, which is somehow in between the two last precited methods, merges the penalty and Lagrange multiplier ideas to eliminate some of the cons aforementioned. Two terms are added to the functional to minimize: one is a penalty term and the other mimic the Lagrange multiplier. However, it does not introduce any additional unknowns because this multiplier is known at each iterations and evolved, in some way, between two of them. In addition the penalty parameter can be significantly smaller than in the penalty method. This method has been applied to the obstacle problem in Auricchio and Sacco (1996).

Shape and topology functional

The second approach expresses the conditions (5.1) to (5.7) under an equivalent functional to minimize, which depends of the shape and topology of the contact zone. The functional is constrained by the plates equations on the whole domain. Then the topology and the shape of the contact zone is evolved until the minimum is reached. This makes use of tools such as the shape, and possibly topological, derivatives to compute a descent direction, to evolve the shape of the contact zone. This method is in the same idea as the one used in the one dimensional case, in the first part of this thesis. It has been dropped in profit of a method which is less complicated to implement. Indeed, in order to implement this method, a tracking of the boundary of the contact zone has to be done. Then a proper numerical method has to be used on each of these subdomains to solve the equations of the plate and the descent direction has to be computed to evolve the interface, until convergence is reached. The method has been found in the literature for the study of the membrane obstacle problem in Hintermüller and Laurain (2010). This formalism can be seen as an active set method, in which the sets are given in terms of curves where contact occurs.

Other approaches

A third approach is to use variational inequalities to formalise it and has been firstly used for Signorini problem with respect to the membrane problem. However, it seems that according to Fukushima (1992), variational inequalities are equivalent to an optimization problem. This approach has been used in Yau and Gao (1992), Fernandes et al. (2001) for von Karman plates and Kirchhoff plates and could be placed in the first category, the optimization problems with functional based on elastic energy.

5.2.2 Discretization

Spatial discretization of the problem is an important feature in solving obstacle problem. Many papers in the literature are based on finite element discretization. This is not surprising as finite element method for elliptic problems are completely equivalent to a minimization problem. In addition, the huge community working on the development of finite elements has developed many algorithms for efficient meshing or for solving systems, but also on mathematical results for error estimates or of various elements, conforming or not. Moreover, the low regularity of the solution makes of the weak form of the equations an interesting formulation and the finite element a proper discretization.

Some papers dealing with the obstacle problem of membrane use the strong form of the equations. In these papers, finite differences are used for the discretization of the whole domain without explicitly treating the interface. However, for the plate obstacle problem the solution respects the strong form only in a piecewise fashion, meaning that it can not be used without explicitly treating the contact boundaries. If no paper dealing with plate obstacle problem using strong form of the equations has been found, some discretization method able to capture discontinuities do exist. These methods could be used to discretize the equations in conjunction with an explicit discretization of the contact interface. Such methods are, for example, finite difference with a special treatment of the boundary to impose the boundary conditions either using ghost points, such as in Coco and Russo (2013) or by changing the stencil and the formula. The computation can be made on a fixed cartesian grid where only the stencils near the interface are modified after the interface has been moved or different grids can be used. Another method is the explicit jump immersed interface Wiegmann and Bube (2000), where the jumps in the function and its derivatives across the interface are added as unknowns and linked to the physics. This is only a small subset of the various numerical methods that can be used to solve the strong form of the plate equations. In addition to this discretization for the plate equations, a tracking of the interface has to be used. Two different approaches can be emphasized here, the explicit one, where the interface is discretized using marker points, and the implicit one, such as in level-set, where the interface is represented by the zero level of a function which is defined in a higher dimension, i.e in 3D for a 2D interface. The latter has the advantage that topological changes are easily handled. This approach has been used for shape and topology optimisation, such as in Sethian and Wiegmann (2000), where the level-set method is used in conjunction with the explicit jump immersed interface method.

5.3 Description of the adopted method and results

Two ideas have been considered to handle contact. The first one was to use the shape and optimization approach with an explicit jump immersed interface method to apply the strong form of the equations. The second one was to use a finite element in conjunction with an active set method such as in Fernandes et al. (2001) and Hartmann et al. (2007).

Comparing both approaches, the second method seems much more simpler to implement. Indeed the first one requires a tracking of the boundary. This means additional computational cost because the interface has to be evolved. Thus, a proper vector field, according to the underlying physics, has to be computed to propagate it. In addition, the solution to the plate equation has to be computed on each sub-domain using the strong form. Using a weak form over the whole domain, without an explicit tracking of the boundary, seems less expensive but, in order to obtain a good result, a fine mesh has to be used at least near the contact interface. So the various possibilities, for computing the solutions and the constraints on the grid, do not allow to affirm that a method is much more computationally intensive than another. At least the one without interface tracking is less error prone due to its lower complexity to implement it. In addition, this approach has already been used in Fernandes et al. (2001), proving its ability to cope with the contact constraint.

5.3.1 Primal-dual active set strategy

In this thesis, a primal-dual active set strategy, as the one used in Hartmann et al. (2007), is added to the numerical scheme, presented in the last chapter, to take into account frictionless unilateral contact constraint. Active set is a rather simple idea, as already introduced earlier, where each node susceptible to come into contact is placed in a set $\mathcal{N}$. This set is divided in two. A set called active set $\mathcal{A}$, where the node is in contact, so the displacement w is known for these nodes, since it is imposed by the position of the obstacle, which is known a priori. Then the force to maintain this node in contact is computed. The other set is called the inactive set $\mathcal{I}$. In this case the counter pressure due to the contact is set to 0 and the displacement is computed using the numerical scheme presented in last chapter. Then (5.1) and (5.2) are verified on the two sets and points which does not respect the conditions are switched to the other set. Using this approach, each node is either in one set or the other. This means the complementarity condition is automatically satisfied.

In order to apply the active set strategy to the plate equations, where dynamics is considered, the constraints have to be cast into a form that involves the incremental displacement $\delta \mathbf{d}$. Since only small displacement and bending are considered, only

the incremental displacement in $\hat{z}$ has to be considered.

$$\delta w_n + w_n \geq h(x, y) \quad (5.9)$$

The contact pressure can be seen as a contribution to the load term p in (4.14), which means in the discrete formulation

$$\int_{\Omega} p_c \tilde{w} d\Omega \approx \int_{\Omega} \tilde{\mathbf{W}} \Psi p_c d\Omega = \mathbf{R}_c^*. \quad (5.10)$$

where $\mathbf{R}_c^*$ is the contact residual vector. It can be seen as the residual of the displacement equations for the discrete value of w . Considering the equations weighted by $\tilde{\mathbf{W}}_1$ only, the residuals, $\mathbf{R}_c$, are an image of the contact pressure, weighted by a positive function, and are used in the algorithm as an image of the Lagrange multiplier. Instead of writing the full system, it is easier to separate the unknowns according to the various sets. Let L_s be the set of all indices in the vector displacement $\mathbf{d}$ for which the displacement u_0 , v_0 or w is not fixed by a boundary condition. This set is composed of L_1 , L_2 , L_3 , L_4 , L_5 and L_6 , the sets of indices with respect to u_0 , v_0 , w , $w_{,x}$, $w_{,y}$ and $w_{,xy}$ respectively. In all these variables only w is constrained so let divide once more L_3 in L_3^A and L_3^I , the sets where the constraint is active i.e. the plates is in contact and inactive, respectively. Based on these subsets, the system (4.90) can be solved to get $\delta \mathbf{d}_n$ on L_3^I . Then the contact residual can be obtained for L_3^A using the relation

$$\mathbf{R}_c = \mathbf{g}_n(L_3^A) - \mathbf{K}_{\text{eff}}(L_3^A, L_s) \delta \mathbf{d}_n(L_s) \quad (5.11)$$

At time step $n + 1$, the active set is initialized to the converged active set of previous step n . Then the active set loop is started and the plates equations are solved on set $\mathcal{J}$ for the variable W and on whole domain for $W_{,x}$, $W_{,y}$, $W_{,xy}$, U_0 and V_0 . A weighed value of the counter pressure p_c is computed for the nodes in the set $\mathcal{A}$, which is simply taken as the residual of the plate equations, $\mathbf{R}_c$. Nodes are then moved from one set to the other. The k nodes with minimal value for $w - \Psi$ in the inactive set and minimal value for the weighted value of p_c in the active set are first searched. Then if they do not respect the inequations (5.1) and (5.2), they are moved to the other set and it goes to next iteration. If the sets do not change in two iterations of the active set loop, the system is considered to have converged to the proper contact zone. Discrete values for displacement, speed and accelerations at time $n + 1$ are then computed and stored, before going to time step $n + 2$. This can seem to be a computationally expensive method but, since the sets do not change a lot between two time steps, it stays affordable. Usually, 1 to 6 iterations of the active set loop is enough, depending on the time step value. The number of nodes that can be moved in one iteration of the active set loop, k , is not restricted to 1 and numerical tests have shown that it can be increased, reducing the computational costs. However changing all nodes which do not respect the inequalities in one iteration usually leads to

convergence issues, or at least increase the number of iterations of the active set loop. A summary of the algorithm is provided in 1.

```

begin
  Compute and assemble the  $\mathbf{K}_{\text{eff}}$  matrix
  Initialize the active sets  $\mathcal{A}$ ,  $\mathcal{A}_{\text{old}}$  and the condition  $\text{cont} = \text{true}$ 
  for  $t \in [t_0, t_{\text{end}}]$  do
    Compute  $\mathbf{g}_n$ 
    Set  $\mathcal{A} = \mathcal{A}_{\text{old}}$  while  $\text{cont}$  do
       $\mathbf{A} = \mathbf{K}_{\text{eff}}$ 
       $\mathbf{b} = \mathbf{g}_n$ 
      if  $\mathcal{A} \neq \emptyset$  then
        | Update  $\mathbf{A}$  and  $\mathbf{b}$  to enforce  $w = \Psi$  on  $\mathcal{A}$ 
      end
      Solve  $\mathbf{A} \delta \mathbf{d}_n = \mathbf{b}$ 
      Compute the residual  $\mathbf{R}_c$  and the gap  $w_n - \Psi$ 
      for  $k = 1 : p$  do
        Find minimal value of the residual  $p_{\min}$  and of the gap  $dg_{\min}$ 
        if  $dg_{\min} < 0$  then
          | Set the value of the gap at this index to zero
          | Set this point in  $\mathcal{A}$ 
        end
        if  $p_{\min} < 0$  then
          | Set the value of the residual at this index to zero
          | Remove this point from  $\mathcal{A}$ 
        end
        Increment  $k$ 
      end
      Set  $\text{cont} = (\mathcal{A} == \mathcal{A}_{\text{old}})$ 
    end
    Compute and store  $\delta \mathbf{d}_{n+1}$ ,  $\dot{\delta \mathbf{d}}_{n+1}$  and  $\ddot{\delta \mathbf{d}}_{n+1}$ 
  end
end

```

Algorithm 1: Primal-dual active set strategy for frictionless contact constraint handling.

5.4 Numerical Study

In order to verify the proposed contact formulation and implementation, the method is applied to the study of the same monomorph as used in the previous chapter, but with a flat obstacle at $z = -50 \mu\text{m}$, and compared to the results obtained in 3D from

Comsol Multiphysics. However, in the latter, the contact constraint is replaced by a spring like force, which is applied only if contact occur and is given by:

$$p_c = k_s |w - \Psi| \text{step}(-(w - \Psi)) \quad (5.12)$$

where k_s is the spring stiffness and the function step is a smoothed version of the Heaviside function. With this type of formulation the penetration of the plate in the obstacle is unavoidable. Penetration can be reduced by increasing the stiffness but for high value, the system is unstable. So the result should be comparable at high value but a difference should be observed. The time stepping normally used by Comsol is adaptive so it will change to capture high frequency component. Since the spring force adds a stiffness in the system the time step can become very small with increasing value of the parameter. So to compare the models the time stepping method is switched from ‘adaptive’ to ‘fixed’ and put to the same value as the one used for the proposed formulation. The time evolution of the displacement at the tip of the bender is shown in fig. 5.2. A fairly good comparison is obtained.

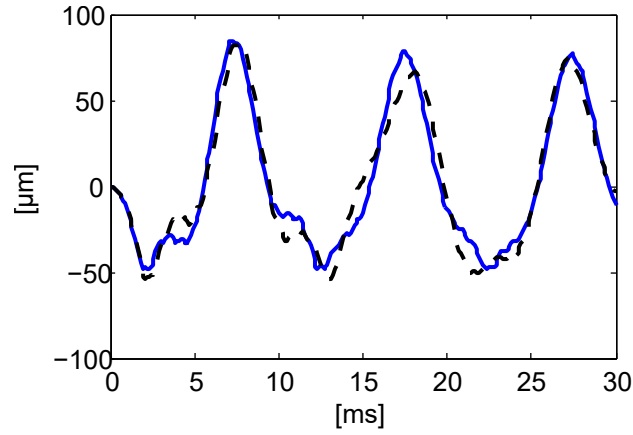


Fig. 5.2: Displacement w at 100Hz with a plane obstacle at $50\mu m$ below the plate. The blue line is the current method. The black dashed line is the 3D Comsol study.

Even if convergence is guaranteed only for permutation of one node per iteration from one set to the other, see Fernandes et al. (2001) and reference there in, numerical experiences performed showed several nodes can be permuted in one iteration. It has been tested up to 50 permutations from both sets with a mesh of around 6000 nodes without showing any convergence issue. This can help to reduce considerably the computational cost of the method.

5.5 Conclusion

In this chapter, the frictionless contact between a thin plate and an obstacle has been investigated. It has been explained that the solution to this problem can not be modelled on the whole domain using the strong form of the equations, due to the discontinuity in the shear force across the contact boundary, at the interface between a part of the domain which is in contact and another which is not. This explains why, in the literature, the weak form of the equations is systematically used.

A comparison of the various approaches encountered in the literature for the frictionless contact of plate, also known as the plate obstacle problem, the signorini problem, the mixed complementarity problem or the free boundary problem, has been done. Then an algorithm based on the primal-dual active set method, adapted to the dynamics of the plate, has been explained and tested in the case of a monomorph coming in contact with a flat obstacle. The results are close to the one observed using a 3D model in Comsol multiphysics with a penalty approach, which helps to be confident with the selected approach.

Application to micropump concepts

6

In this chapter, the developed equations for dynamics of coupled bending-extension of laminated plates are applied to the two micropumps concepts. The results are compared in terms of displacement, stresses and electric field to the ones obtained from a 3D model developed in Comsol Multiphysics, showing the good agreement obtained, as well as the huge saving in computational time.

The first section is dedicated to the electromagnetic micropump and the second section, to the piezoelectric one. In each of these sections, the 3D model developed in Comsol Multiphysics is firstly explained. This is followed by the explanation on the laminated plate model. Then numerical results are given, comparing both models in terms of results, computational cost and memory requirement.

6.1 Electromagnetic micropump

In order to evaluate the performance of the laminated plate theory to model the electromagnetic micropump, it is compared to a 3D model developed in Comsol Multiphysics, which will be given with the adopted simplifications in the following, to show the good agreement between both models.

6.1.1 3D model

Modelling the electromagnetic micropump, even without considering the fluid, is quite challenging because of the coupling between the two physics, electromagnetism and structural dynamics, as well as the contact. In addition, the high aspect ratio in the geometry requires special care in the meshing procedure and in the modelling process. Indeed, the diaphragms are tens of millimetres large and long for a couple hundreds of micrometres thick, where several layers of different materials are used. In addition, the diaphragms move in an air gap a couple of millimetres high and as

large and as long as the diaphragms, where a magnetic field is generated by permanent magnets. Meshing such a geometry is quite challenging, especially with automatic tools.

In order to simplify the task, some assumptions are introduced. As in the first part of this thesis, the magnetic field is computed taking into consideration only the permanent magnets. This allows to model the micropump as two parts, called component in Comsol software, with different geometries and only one of the physics for each, which are coupled through an extrusion coupling operator.

The first component: magnetic field computation

The geometry of the first component, shown in fig. 6.1, uses the two symmetry planes xz at $y = 0$ and xy at $z = 0$ to reduce the computational cost and memory requirement. It is composed of several rectangular parallelepipeds, where the ones with number 1 to 15 represent the permanent magnets, the 16 is the air gap, the 17 and 18 are their to represent the surrounding environment and the 19 is the iron core on which the magnets are fixed. The core has been added, compared to the design in the first part of the thesis, to help mounting the magnets in the prototype and should be taken into account in the modelling. Domains 1 to 18 are all modelled using air as material. This is not a big assumption for the magnets since their magnetic permeability is nearly the same. The core is made of soft iron, and modelled without loss as defined in the Comsol library.

The physics interface used is the magnetic field one, which solves the magnetic potential vector A :

$$\nabla \times \mathbf{H} = \mathbf{J}_e \quad (6.1)$$

$$\nabla \times \mathbf{A} = \mathbf{B}, \quad (6.2)$$

where $\mathbf{J}_e$ is the current density load, which is null in this case. Comsol uses quadratic polynomials to represent the potential vector on the element. Constitutive equations for magnets are defined using the remanent flux density option of the Ampere law, one for each polarization direction. Magnetic insulation, i.e. $\mathbf{n} \times \mathbf{A} = 0$ is used to impose the symmetry condition at the two planes, xz at $y = 0$ and xy at $z = 0$. Magnetic potential vector is imposed to 0 on the 4 other exterior boundary faces. A general extrusion operator is used to have access to the value of magnetic field, computed in the air gap, in the second component of the model. The map used between the source and destination domains is given by:

$$(x, y, z) \longrightarrow (x, y, |z|). \quad (6.3)$$

The absolute function is used in order to allow Comsol to compute the field at a negative value in z , which express the fact that the field is symmetric with respect

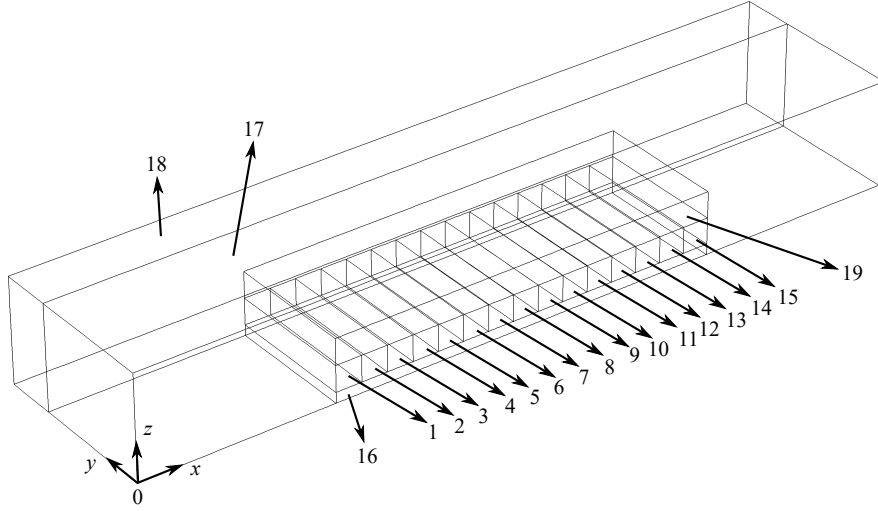


Fig. 6.1: Geometry for magnetic field computation, where planes xz at $y = 0$ and xy at $z = 0$ are symmetry planes. The domains with number 1 to 15 represent the permanent magnets, the 16 is the air gap, the 17 and 18 are their to represent the surrounding environment and the 19 is the iron core on which the magnets are fixed.

to the xy plane at $z = 0$. This extrusion operator will be necessary to compute the loading term, in both upper and lower plate, in the second component.

The mesh, shown in fig. 6.2, is built considering that magnetic field does not vary a lot along the y direction on the length of the magnet. Thus, a triangular mesh is built on the face xz at $y = 0$ and then extruded in the y direction, on domains 1 to 17, as well as on the 19 one. On domain 18, the mesh is made of tetrahedral elements.

The second component: structural mechanics

The geometry for the second component also use the symmetry to reduce the computational load. The xz plane at $y = 0$ is used to model only one half of the two plates. Each plate is composed of a layer, made in kapton, on which 5 copper tracks are fixed and then a second layer of kapton is placed, on the whole plate, to insulate the copper tracks. The segmentation gap, between each copper track, is null here. The second plate is just the same, mirrored with respect to the xy plane at $z = 0$. The plate geometry is shown in fig. 6.3.

The physics interface used here is the structural mechanics, which solves:

$$\rho \frac{\partial^2 \mathbf{u}}{\partial t^2} - \nabla \cdot \mathbf{T} = \mathbf{F}_v, \quad (6.4)$$

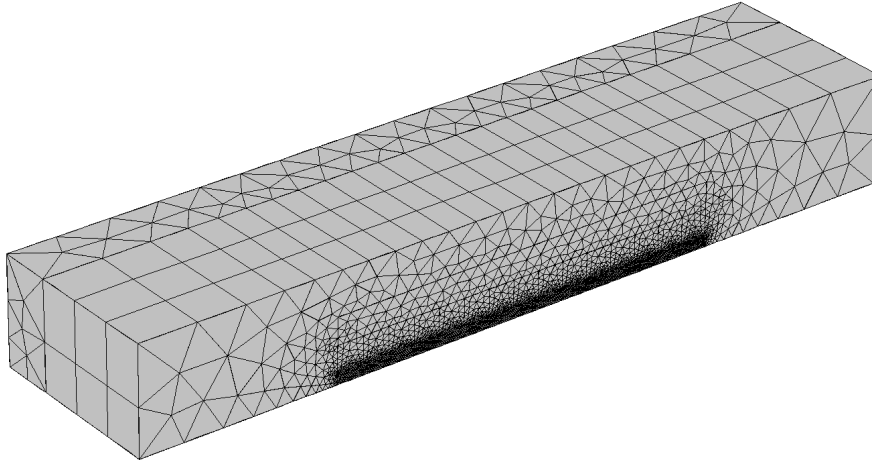


Fig. 6.2: Mesh for magnetic field computation.

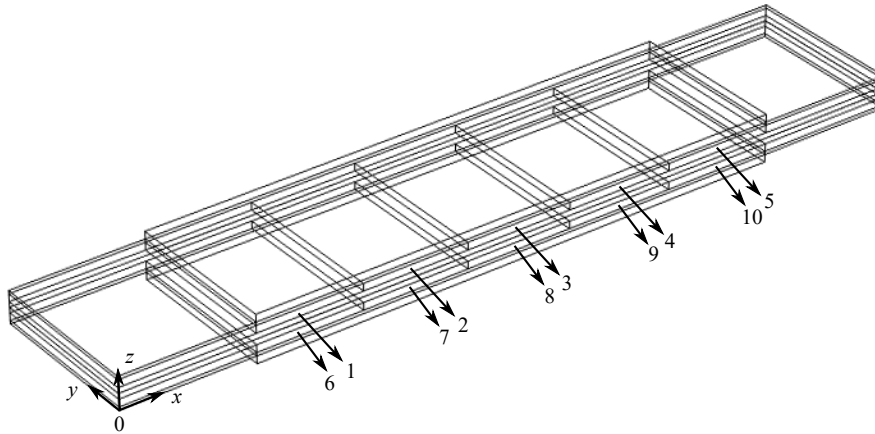


Fig. 6.3: Geometry for structural dynamics study, where domains numbered 1 to 10 are the copper tracks, the other are made in kapton. It has been magnified 10 times in the thickness direction.

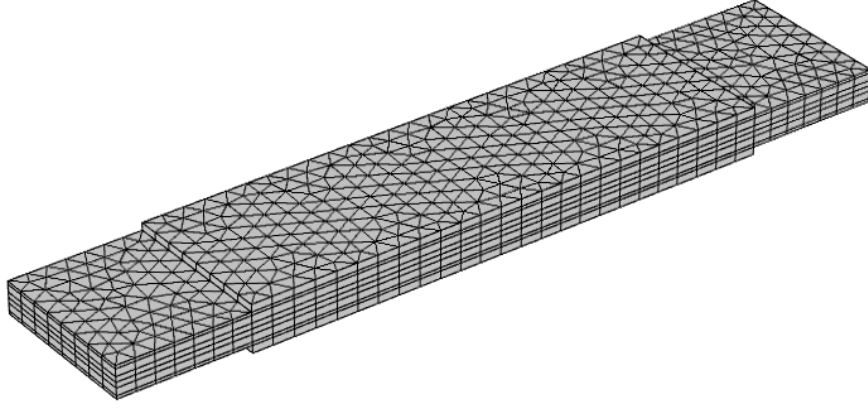


Fig. 6.4: Mesh for structural dynamics study. It has been magnified 10 times in the thickness direction.

where $\mathbf{u}$ is the displacement vector, $\mathbf{T}$, the stress tensor and $\mathbf{F}_v$, the force density. The problem is formulated in terms of displacement which are discretized using cubic polynomials. The constitutive equations for the materials are linear elasticity. A symmetry condition is used on all the faces in the xz plane at $y = 0$ and a clamped condition is imposed on the three other boundaries of the plate, using the rigid connector node for faces, where the three displacement components and the two rotations with respect to x and y axis are constrained to 0. Each copper track is loaded independently by a force density which represents the Lorentz force. The force is due to the current density, and to the magnetic field, which is computed using the general extrusion coupling operator defined in the first component, computed as in (6.7). It is introduced in the model using a volume force density node, where both force densities in z and x directions are implemented. The contact constraint is handled using the contact pair node provided by Comsol and which uses an augmented Lagrangian formulation. Compared to the 3D model used previously, this formulation has been preferred to the penalty one because some convergence issues have been observed for this application. From the observation made, it was concluded that the penalty also introduces an adhesion like effect, which leads to a converged solution where the displacement was under estimated.

The mesh is built considering that the displacement will not vary a lot with respect to the thickness coordinate. Thus a 2D triangular mesh is built in the plane xy which is then extruded in the z coordinate with only one layer of elements, giving the mesh shown in fig. 6.4.

Study and solvers

The model has been used to analyse the pump at steady state, when three phase are powered together. The solution is computed in two steps, using two study nodes, in order to reduce the computational load. The first one is a stationary step where only the first component is used to compute the magnetic field. All the variables for structural mechanics are put aside. An iterative solver, GMRES, is used to compute the solution. The second step is also a stationary study, where only the structural mechanics of the plates is solved. It uses the computed magnetic field at the first step to evaluate the Lorentz force density acting on the plates, taking into account the variation of the magnetic field in the air gap. Working this way helps to reduce the computational load, especially in case of time dependent study, since the magnetic field, which is not impacted by the displacement of the plate, is computed only once instead of at each time step. The solver uses a segregated step where all the variables, except the contact pressure, are solved in a first time, and then these values are used in a second time to compute the contact pressure. The MUMPS direct solver is used to compute the systems obtained in these two steps.

6.1.2 Laminated plate model

Compared to the previously presented model in chapter 3 some assumptions have been dropped. Once again the magnetic field for Lorentz force density computation considers magnets' contribution only. The magnets are now fixed on an iron core which is supposed to be infinite in the z direction, above the magnets but also in the y direction as for the magnets. Considering this the magnetic flux density in x direction is given by:

$$B_x = \sum_{m=0}^{\infty} a_m (\cos(\lambda_m x) \cosh(\lambda_m z) \hat{x})$$

in which, if $\mu_{r2}/\mu_{r1} \ll 1$ is verified:

$$a_m = \frac{(-1)^{m+1}}{(2m+1)\pi} \frac{4B_r}{\cosh(\lambda_m(e + H_m))} \left[(\cosh(\lambda_m H_m) - 1) \cos\left(\lambda_m \frac{L_V}{2}\right) + \sinh(\lambda_m H_m) \sin\left(\lambda_m \frac{L_V}{2}\right) \right] \quad (6.5)$$

$$\lambda_m = \frac{(2m+1)\pi}{L_V + L_H} \quad (6.6)$$

which can be integrated to find the magneto-mechanical coupling term. This one is nonlinear with respect to w and given by:

$$\begin{aligned} \mathbf{p}_{ED} &= \sum_{k=1}^N \int_{w+h_{k-1}}^{w+h_k} J_k \hat{\mathbf{y}} \times (B_x \hat{\mathbf{x}} + B_z \hat{\mathbf{z}}) \\ &= \sum_{k=1}^N \int_{w+h_{k-1}}^{w+h_k} -J_k B_x \hat{\mathbf{z}} + J_k B_z \hat{\mathbf{x}} \end{aligned} \quad (6.7)$$

In these force densities, only the one exerted in $\hat{\mathbf{z}}$ is kept. The coupling is one way, since the currents flowing in the plate are supposed not influencing the magnetic field. In the model developed in chapter 3, the Lorentz term was supposed to be constant and evaluated at $z = 0$. Here the evolution of the force density with respect to the displacement of the diaphragm is taken into account in a loose manner. It is computed with respect to the position of the diaphragm at each time step and is considered to be constant over the incremental displacement δw . The non linear set of equations, resulting from the non linear pressure load, is thus linearised by this approach. For the stationary study, a transient is also computed, until the steady state is reached.

For the structural dynamics, the plate equations and the previously presented numerical scheme with contact constraint handling are used. The copper is supposed to be continuous between each electric track i.e. the discontinuity in displacement imposed by the separation between each track is not modelled.

This model is used to compute the displacement, the volume of the bubble and the stresses in the thin plate. For this analysis, the thin plate is considered to be made of a first layer of kapton, a second layer with electric path and a third layer of kapton to ensure electrical insulation. These layers are respectively 50, 75 and 100 μm thick. The plate is 30 mm wide. Permanent magnets in Neodyme are used. A constant current density, arbitrarily set at 5 A mm⁻², circulates in three electric tracks.

6.1.3 Numerical results

A first analysis is conducted to show the displacement and the expected von Mises stress in the plate, at steady state. These ones are taken at the lower surface of the plate. Fig. 6.7a show the bubble form, which is what was searched. Fig. 6.5a and fig. 6.6a show that u_0 and v_0 remain small compared to w , since they are around 1 percent of w . von Mises stress shows that they remain quite below the yield strength which is around 60 MPa for kapton. This was done for an air gap of 1 mm and with magnets of 5 by 5 mm.

In order to verify some of the assumptions, the displacement and von Mises stress are compared to the ones obtained with the 3D model developed with COMSOL

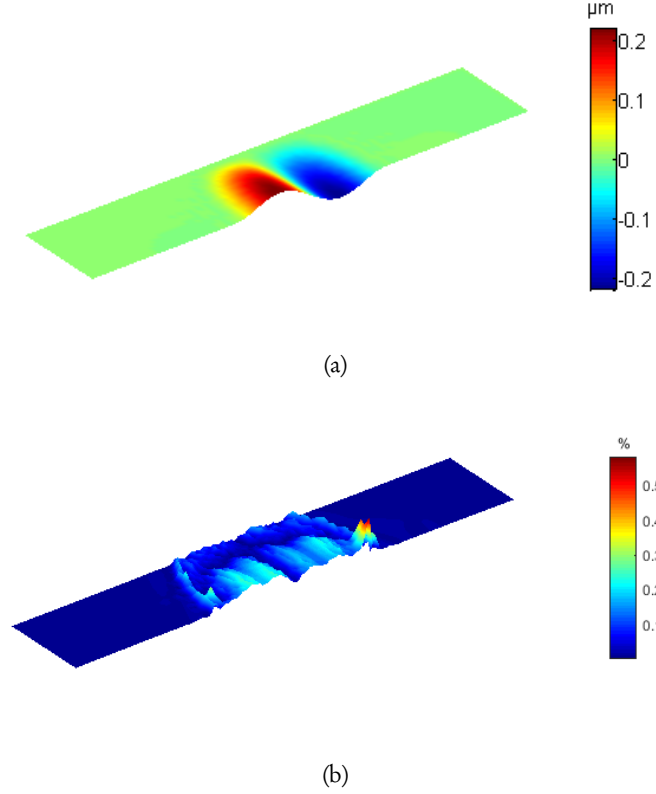


Fig. 6.5: u_0 , computed using the laminated plate model (a), and its local relative difference with respect to its maximum absolute value over all the plate (b), compared to the 3D Comsol model for the electromagnetic micropump.

Multiphysics. The relative difference has been computed in relative L^2 -norm given by:

$$|\mathbf{E}|_2(u) = \sqrt{\frac{\sum_{i=1}^N (U_i - U_i^h)^2}{\sum_{i=1}^N U_i^2}} \quad (6.8)$$

This difference has been computed using interpolated data on a grid. The relative difference for u_0 , v_0 , w and von Mises stress are respectively 0.28, 0.57, 0.27 and 1.5 percent. It has also been computed for the volume which is of 0.19 percent. The relative difference has also been computed locally to show its repartition on the domain. It is shown in 6.5b, 6.6b, 6.7b and 6.8b. Displacement components are in good agreements, but it is less the case for von Mises stress, where bigger local discrepancies are found around the contact zone. This is not surprising since at the contact line concentrated loads appear to satisfy equilibrium. These modify the stress state which is

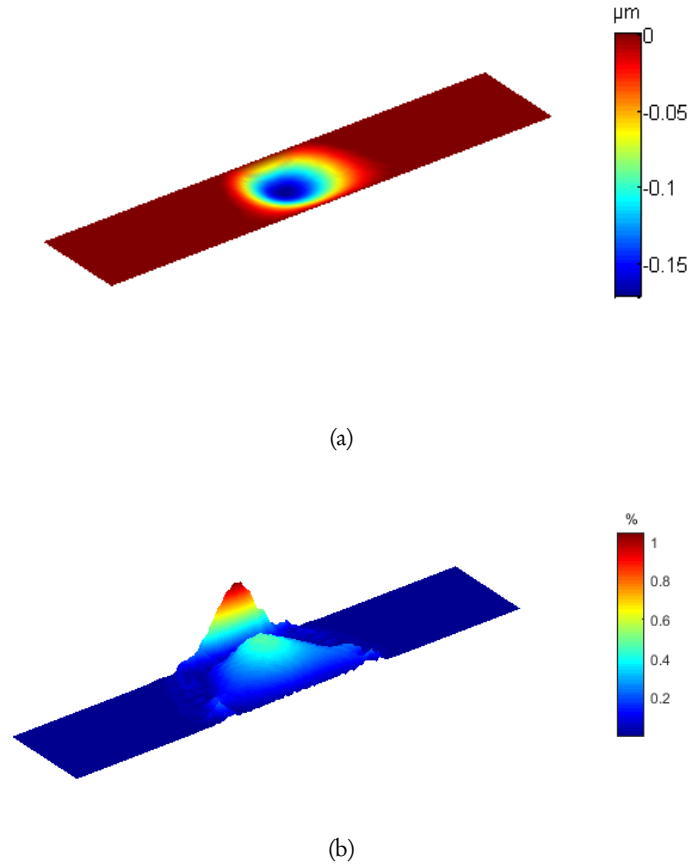


Fig. 6.6: v_0 , computed using the laminated plate model (a), and its local relative difference with respect to its maximum absolute value over all the plate (b), compared to the 3D Comsol model for the electromagnetic micropump.

certainly 3D there. The gain in computational time is huge since it is around 500 s for the plate model with the finer mesh against around 11 000 s for the 3D model. Moreover if the model is run for coarser element size, the computational time can be drastically reduced depending on the expected accuracy. The experience has been conducted for two coarser meshes. The computational times are 2.5 s and 25 s for a difference on the volume of 3.7 and 1.75 percent respectively compared to the finer mesh for the laminated plate model.

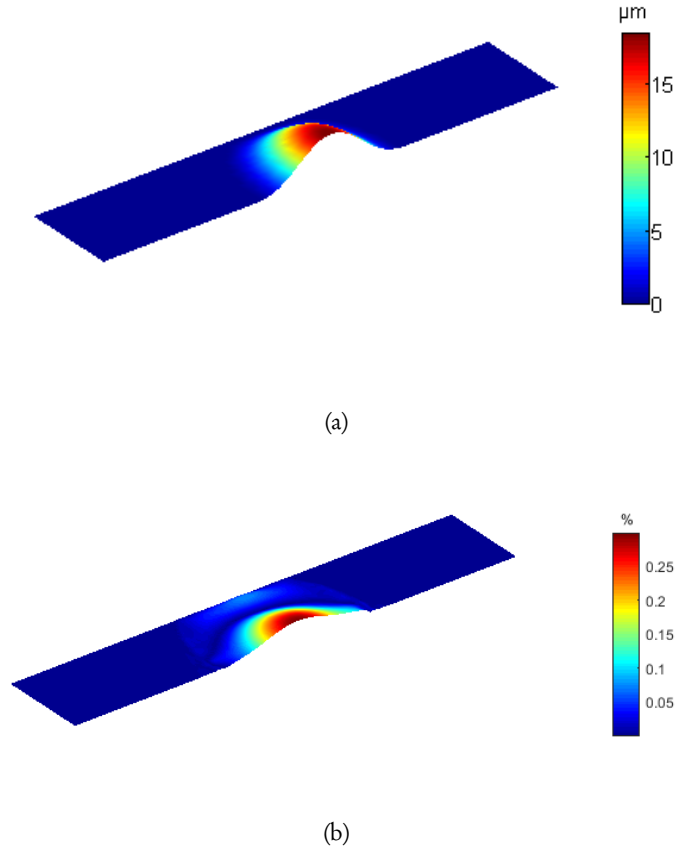


Fig. 6.7: w , computed using the laminated plate model (a), and its local relative difference with respect to the maximum absolute value over all the plate (b), compared to the 3D Comsol model for the electromagnetic micropump.

6.2 Piezoelectric micropump

In order to evaluate the performances of the laminated plate theory in modelling the piezoelectric micropump, it is compared to a 3D model developed in Comsol Multiphysics, which will be given in the following.

6.2.1 3D model

In this case only one physics interface has to be used, since Comsol has already a built-in "piezoelectric device" module, which couples electrostatics and structural dynamics. It is used here to analyse the micropump, considering only one half of it, as shown in fig. 6.9, in order to reduce the computational cost. It is composed

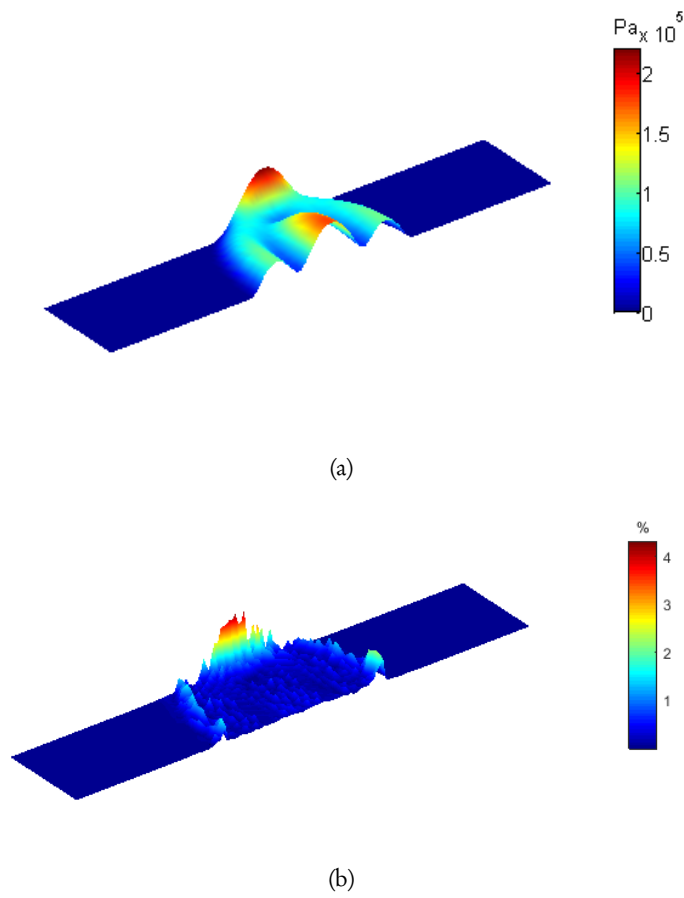


Fig. 6.8: von Mises equivalent stress, computed using the laminated plate model (a), and its local relative difference with respect to its maximum absolute value over all the plate (b), compared to the 3D Comsol model for the electromagnetic micropump.

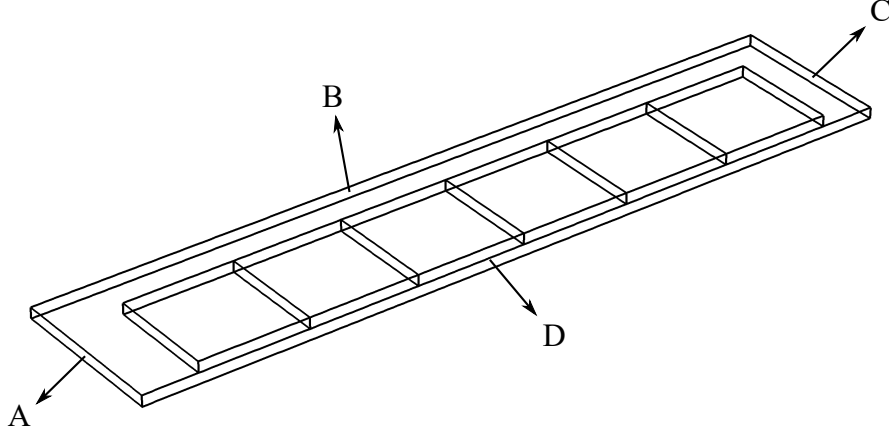


Fig. 6.9: Geometry used to model the piezoelectric micropump. To reduce computational cost, only one half of the pump is modelled and symmetry boundary condition is imposed. In addition the bottom of the micropump is replaced by a contact condition using a penalty approach, which is implemented using a spring like boundary load.

of a 0.5 mm-thick-diaphragm made in AISI 4340 steel, on which a 0.5 mm-thick-piezoelectric patch, made in PZT-5H, is fixed. This patch is divided in 6 domains to impose a different voltage on each of it, but the continuity of the displacement in the patch is maintained. The steel diaphragm is modelled by linear elasticity and the patches by linear piezoelectricity. Displacement, as well as rotation along x and y axes, on faces A, B and C are constrained to 0. The faces in the D plane are the symmetry ones, on which the symmetry boundary condition is imposed. The bottom of the micropumps is not discretized here, a contact condition is used instead and is implemented using a penalty approach, by adding a spring like boundary load. In order to improve the convergence rate, this load p_c is imposed to:

$$p_c = k_s |w| \text{step}(-w), \quad (6.9)$$

where the step function is a smoothed Heaviside function, which will be 0 when there is no contact. In addition, a boundary load is applied on top of the diaphragm and patches, which vary linearly with respect to the x direction. For the piezoelectric patches, their lower faces are put to ground and a voltage is imposed on their upper faces.

The top face of the steel diaphragm is meshed using a triangular mesh, which is then extruded in the thickness direction to mesh all the domains, as shown in fig. 6.10. The triangular mesh is made using the predefined size ‘extremely fine’ for the general physics purpose. The extrusion is made using a distribution of 2 elements, which results in two layers of element, in the thickness direction, for the

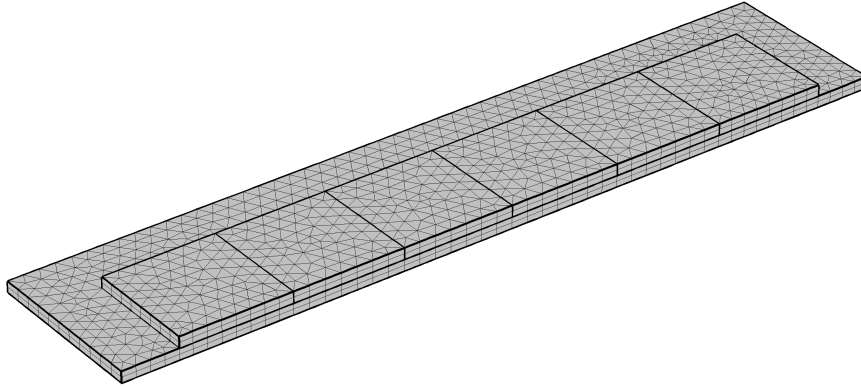


Fig. 6.10: Mesh made of the extrusion of a triangular mesh built on the top face of the steel diaphragm.

steel diaphragm and for the piezoelectric patches.

Study and solvers

The model is used for two different purposes. The first one is the computation of the static displacement of the diaphragm when several patches are powered. It uses a stationary step to compute the displacement, in a fully coupled way, with a Newton method, where each linear system is solved using a MUMPS solver. The second purpose is to compute the dynamics, where two steps are used. The first one is a stationary step, which is exactly the same as the previously explained one, which compute a consistent initial value for the time dependent step. The time stepping method is the generalized alpha method, where the step is fixed. This helps to reduce the computational cost by smoothing the effect of contact condition, which adds high artificial stiffness to the system and thus leads to a small time step, if Comsol can choose freely its time step. At each time step, a nonlinear system is solved, using the Newton method with a MUMPS solver for the formed linear system.

In both cases, the micropump is submitted to an external surface load which varies linearly along the micropump going from 0 Pa to 10 000 Pa. For the stationary study, 3 piezoelectric patches are powered at 200 V. In the time dependent study, a trapezoidal voltage wave form of 200 V is used, with a phase shift such that it is powered during 4 periods, one to go up, two at maximum voltage and one to go down and a given time scale such that in one time scale, the voltage goes from 0 to the full voltage.

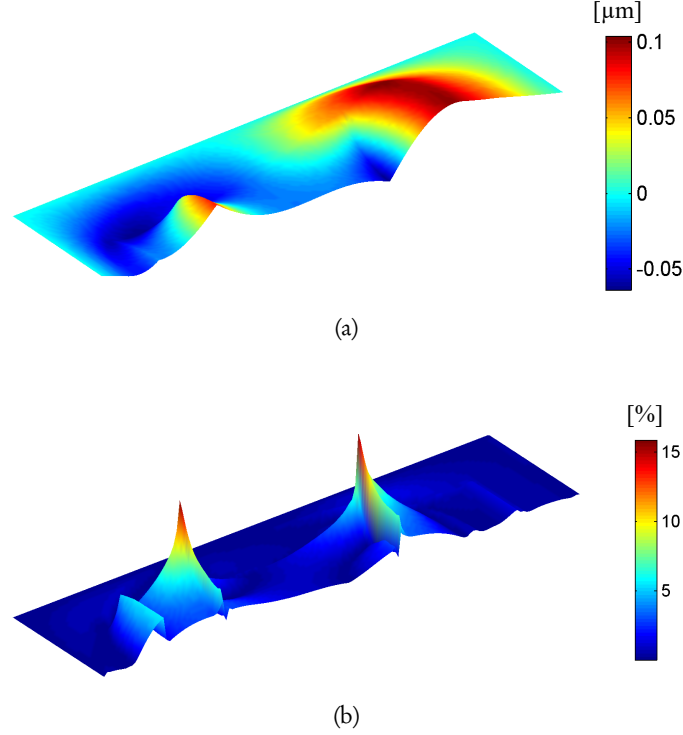


Fig. 6.11: u_0 displacement (a) and its relative difference (b) in the static case study. A relatively good agreement is observed, except near the interface between powered piezoelectric patches and unpowered ones, especially near the corners.

6.2.2 Numerical results

In the steady state case, the displacement components, as well as the von Mises equivalent stress, the electric field, and their local relative difference are shown in fig. 6.11, 6.12, 6.13, 6.14, 6.15 and 6.16. The local relative difference for variable u , $E_1(u)$, is defined as:

$$E_1(u) = \frac{|u_m - u_c|}{\max |u_c|}, \quad (6.10)$$

where u_c is the value given by the Comsol 3D model and u_m is the value given by the plate model implemented in Matlab. To avoid large relative difference where the displacement is nearly zero, but also to measure the difference with respect to the maximum value of the displacement, the absolute difference, between the plate model and the 3D one, is normalized with respect to the maximum displacement computed over the whole domain, rather than the value measured at the point. It shows a good agreement, below the 5 percent, except close to the interfaces between powered piezoelectric patches and unpowered ones, as well as the interface between

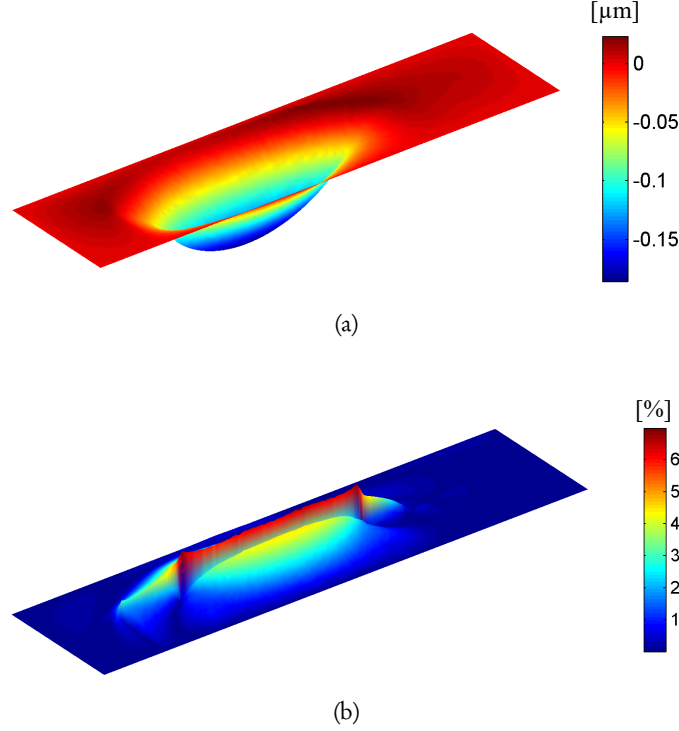


Fig. 6.12: v_0 displacement (a) and its relative difference (b) in the static case study. A good agreement is observed, except near the interface between powered piezoelectric patches and elastic diaphragm.

piezoelectric patches and the elastic diaphragm. In order to verify that, the global relative difference in L^2 -norm, defined by:

$$|\mathbf{E}|_2(u) = \sqrt{\frac{\sum_{i=1}^N (U_i - U_i^h)^2}{\sum_{i=1}^N U_i^2}} \quad (6.11)$$

is computed over the whole plate, but without taking into account points at a distance less than $2h$ from the interfaces, where h is the thickness of the plate. For the displacement components u_0 , v_0 and w , it was 4.22, 7.25 and 3.55 percent, respectively. Another quantity of interest is the volume of the bubble for which the relative difference is 5.18 percent. For von Mises equivalent stress in the elastic layer, it was 5.4 percent and 1.32 percent in the piezoelectric one. It is even better for the electric field where it was only 0.3 percent. These relative differences are higher than the one observed for the electromagnetic micropump. We propose the two following explanations. The first is that the behaviour near the discontinuities is not properly modelled and require higher order terms in the thickness expansion of the displace-

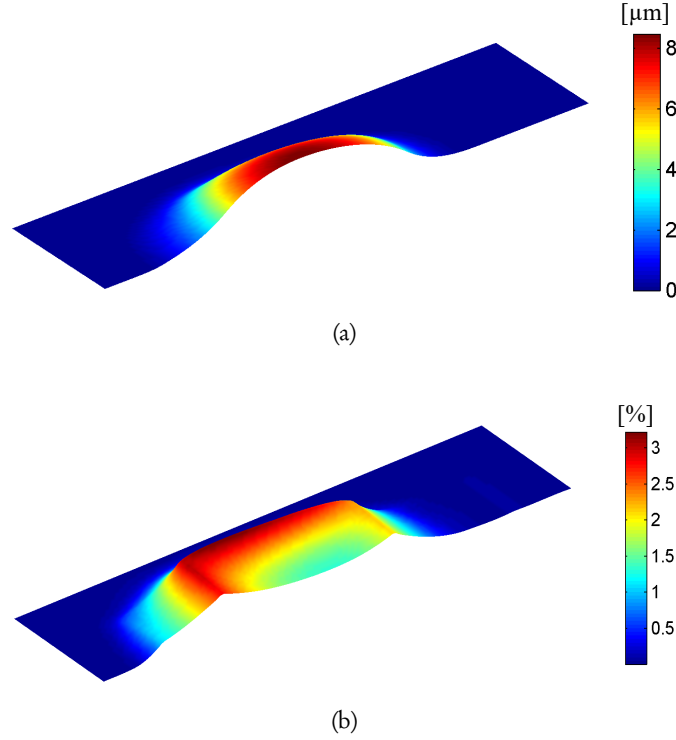


Fig. 6.13: w displacement (a) and its relative difference (b) in the static case study. A good agreement is observed, except near the interface between powered piezoelectric patches and unpowered ones, as well as near the interface between powered piezoelectric patches and the elastic diaphragm.

ment components or even a 3D consideration. This explains, at least partially, the difference which is observed near the interfaces. The second explanation is that the aspect ratio is such that the diaphragm can no longer be considered as a thin plate but rather as a moderately thick plate or even as a thick plate, requiring higher order plate theories. This can explain why the differences, on the whole domain, in the piezoelectric micropump are bigger than the ones observed in the electromagnetic micropump.

In addition to these accuracy results, several things can be said on the computational performance of the models. The first one is over the Comsol 3D model for which three computations have been performed with different values for the penalty parameter, $5 \times 10^{11} \text{ N m}^{-3}$, $1 \times 10^{12} \text{ N m}^{-3}$ and $5 \times 10^{12} \text{ N m}^{-3}$. The maximum value of the penetration inside the obstacle was $0.09 \mu\text{m}$, $0.05 \mu\text{m}$ and $0.02 \mu\text{m}$, for computational times of 89 s, 102 s and 217 s, respectively. This was performed with the mesh shown above. This shows the convergence sensitivity with respect to the

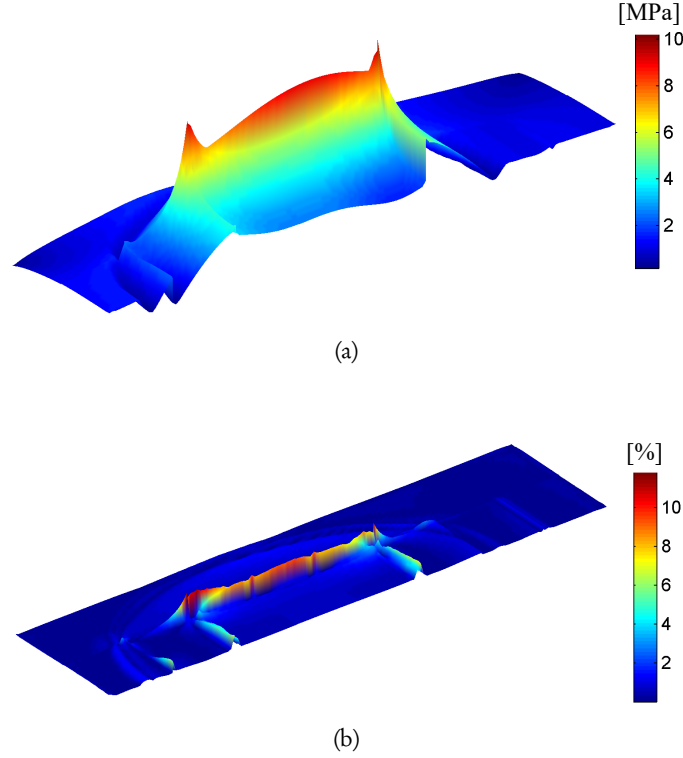


Fig. 6.14: von Mises equivalent stress (a) and its relative difference (b) in the elastic layer and in the static case study. A good agreement is observed, except near the interface between the various subdomains.

spring stiffness, for which the value also depends of the external load.

The Comsol model uses cubic polynomials for displacement discretisation, as well as a penalty parameter of $5 \times 10^{12} \text{ N m}^{-3}$, for a computational time of 990 s and a memory consumption of 5.3Go. In comparison, the model implemented in Matlab has taken around 37 s and around 700Mo for the mesh used for the accuracy test, with a maximum element size of 0.5 mm. In addition, this time can even be reduced more if a small reduction of accuracy is authorized. The global difference, for u_0 , v_0 and w , on 3 meshes, with maximum element size of 2 mm, 1 mm and 0.5 mm, has been computed with respect to a finer one with maximum element size of 0.25 mm for the laminated plate model. As shown in fig. 6.17, differences remain small. The computational times are around 0.6 s, 4 s, 37 s and 428 s, respectively, for a difference of 1.54 percent for u_0 , 1.64 percent for v_0 and 0.28 percent for w between the coarsest and the finest mesh, measured with $|\mathbf{E}|_2(u)$. Comparing the 3D model and the fastest Matlab one, it is a huge gain in computational time and memory requirement, which is useful if the model has to be used in optimization

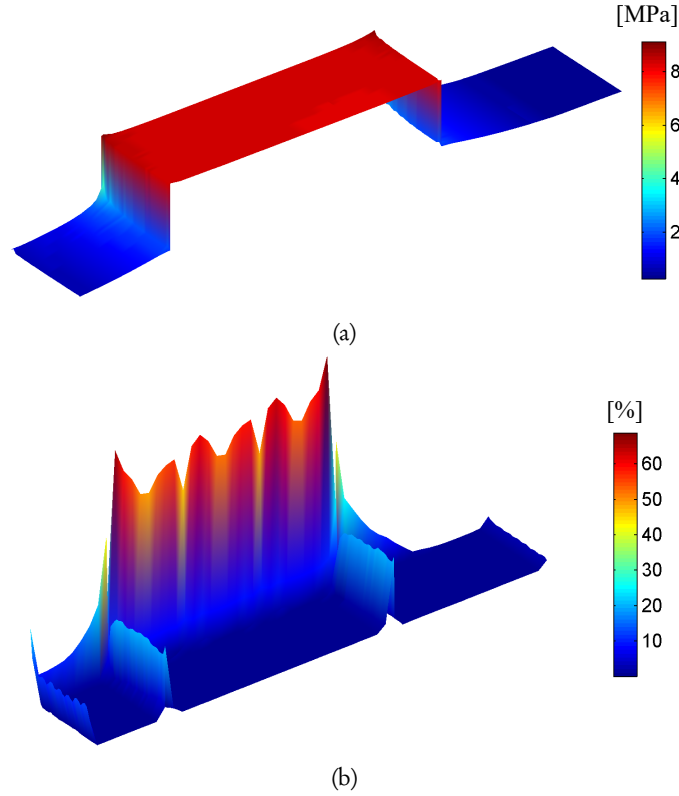


Fig. 6.15: von Mises equivalent stress (a) and its relative difference (b) in the piezoelectric layer and in the static case study. A good agreement is observed, except near the interface between powered patches and unpowered ones, as well as near the boundary with the elastic diaphragm.

process, where the model is evaluated many times.

The models have also been compared in a time dependent study, where the comparison is based on the time evolution of the volume under the diaphragm. It appears that the spring stiffness value can lead to convergence issues in dynamics, even if the stationary step converges to a displacement which is in good agreement with the laminated model. As shown in fig. 6.18 (a) compared to fig. 6.18 (b), a too high value, which is geometry dependent, will influence the solution. It has to be noted that Comsol did not indicate any problem during the computation, such that both solution, represented by the blue curves, have converged. An order of magnitude of difference between both stiffness value leads to quite different results, even if in fig. 6.18 (a), the global behaviour seems respected, it is clearly better in fig. 6.18 (b).

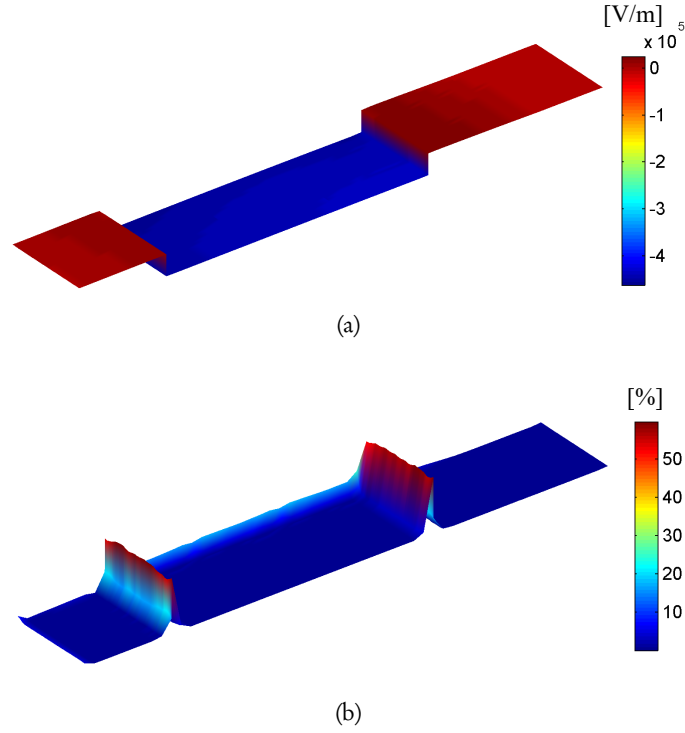


Fig. 6.16: Electric field (a) and its relative difference (b) in the static case study. A good agreement is observed, except near the interface between powered patches and unpowered ones, but also between the elastic diaphragm and the powered patches.

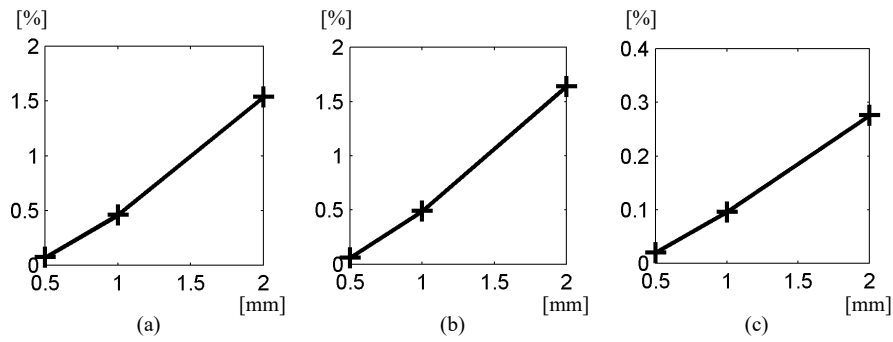


Fig. 6.17: Mesh convergence study for the laminated plate model, showing the difference computed based on $|\mathbf{E}|_2(u)$, for (a) u_0 , (b) v_0 and (c) w , for an element maximum size of 2 mm, 1 mm and 0.5 mm compared to a mesh of 0.25 mm.

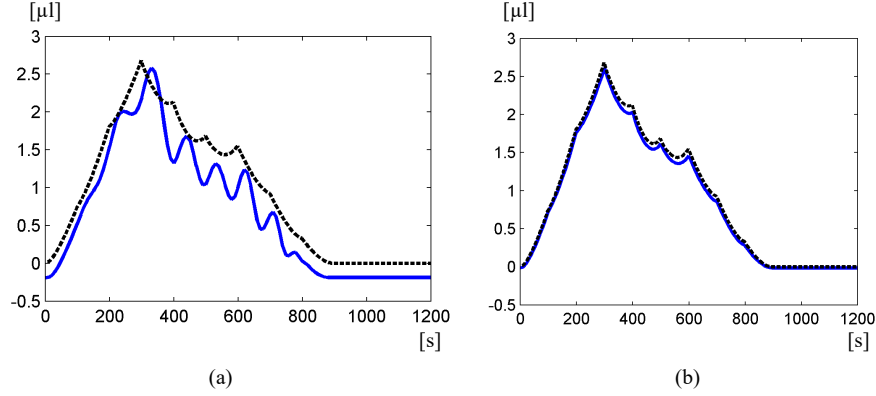


Fig. 6.18: Volume under the diaphragm in dynamics, for 2 different spring stiffnesses. The blue curves are the results of Comsol models and the black dashed ones are from the laminated plate models. One order of magnitude between stiffnesses results in quite different dynamic, showing the sensitivity to the value of the parameter. This is not a problem for the primal-dual active set method which does not require any parameter tuning. The simulation have been performed using a time scale of 100 s.

6.3 Conclusion

In this chapter, the laminated plate equations with contact handling have been applied to model both electromagnetic and piezoelectric peristaltic micropumps. Three dimensional models developed in Comsol Multiphysics software, to compare with the laminated plate ones, have been presented.

A numerical study, at steady state, has shown that displacement component as well as von Mises equivalent stress compare quite well in the electromagnetic pump where local relative difference was under 1 percent for the displacement and a bit more than 4 percent for von Mises stress. In addition, a global difference computed in L^2 -norm was of 0.28, 0.57, 0.27 and 1.5 percent for u_0 , v_0 , w and von Mises stress, respectively, has shown that the model was in good agreement with a computational time of 500 s for the plate model and around 10 800 s for the model developed in Comsol.

A similar numerical analysis for the piezoelectric micropump has shown that the results were in quite good agreement but with local relative difference which are higher, sometimes as high as 15 percent and even more for von Mises stress at the vicinity of the boundary where it was around 70 percent. These high values are however localized around the boundaries and material interfaces where assumptions made over the displacement expression, but also for the electric field study, are no longer acceptable. Global difference in L^2 -norm were of 4.22, 7.25 and 3.55 for

displacement components u_0 , v_0 and w and of 5.4 and 1.32 percent for von Mises equivalent stress in elastic and piezoelectric layers. The observed difference for the electric field was only of 0.3 percent. Once again the laminated plate model as shown a great gain in computational efficiency with a computational time around 0.6 s for the laminated plate model compared to 990 s for the one developed in Comsol software.

In addition, some computations have been performed in dynamics to compare both models. This has been done only for the piezoelectric micropump, since the computational cost for 3D model of the electromagnetic micropump was already prohibitive in the stationary case. It has put forward, once again, the sensitivity of the response to the spring parameter. Even if Comsol does not indicate any convergence issues, computations made for two different values of the stiffness have shown that it has an impact on the solution. This shows that the primal-dual active set method, which does not need any parameter tuning, is much more robust.

Conclusion and perspectives

7

Two original designs of peristaltic and valveless micropumps, where the diaphragm is bent to create a travelling wave and to move the fluid, have been introduced in this thesis. They differ by the actuation used to bend the diaphragm, where the first uses the action of Lorentz force, induced by the interaction of a current density flowing in the diaphragm and of a static magnetic field due to permanent magnets, and the second uses piezoelectric elements, fixed on the diaphragm, to generate an internal bending moment. The rectification is ensured either by the Lorentz force or by an externally applied pressure, depending on the design, which leads to micropumps where diaphragms are in contact and avoid a back flow to appear.

Performing simulations helps to understand how the devices work and allow to evaluate performances, such as flow rate but also things that can not be measured directly such as the maximum value of the stresses in the material. However, modelling such devices is not an easy task because it involves structural dynamics of the diaphragm, with in addition mechanical contact constraint, coupled to fluid mechanics for the fluid flow but also magnetostatic or electrostatic, depending on the actuation. In addition, this model has to be robust, computationally efficient, accurate and sufficiently complete to represent the underlying physics. A trade off appears here between its complexity and completeness on one side and its computational cost, which must be reduced to authorize its intensive use in optimization process. Fortunately, both designs have many in common, meaning a unified framework can be developed to model both micropumps.

This thesis is dedicated to this aim: providing a tool to model devices involving the dynamics of a diaphragm composed of several layers where mechanical contact can occur. It has to be sufficiently accurate to evaluate the stresses, the electric field at a reasonable computational cost. Both micropumps are used as benchmark case to evaluate the developed tool.

For this purpose, a first attempt has been made where the diaphragm is thought as a one dimensional object and, using Euler-Bernoulli assumptions, a static laminated beam theory, which includes loads due to piezoelectric effect or Lorentz force, has been developed. The effects of the fluid are completely neglected in this case. It has been used to compute the static deflection, with contact boundary conditions, of the micropumps, showing its limits. It has been noted that the model failed to evaluate properly the stresses in the width direction of the beam, but also its deflection, when higher external loads are applied. In addition, the piezoelectric effect in the width direction of the beam, the boundary conditions, as well as the mechanical contact that can occur in the width direction of the beam, can not be properly taken into account with this approach. However, it has the advantage that an analytical solution can be obtained. This was the subject of the chapter 2 and 3, with general conclusion that a more evolved theory has to be considered.

This was the subject of the second part of this thesis, where a laminated plate theory, based on Kirchhoff-Love assumptions and which includes loads due to piezoelectric effect and Lorentz force, has been developed. The electrical equations, for piezoelectric materials, are solved analytically, leading to an expression for the electric field which is dependant of the applied voltage but also of the curvature of the plate. This helps to reduce the computational cost since electrical variables, which are usually solved numerically in commercial software, are replaced by an equivalent loading term and an equivalent stiffness coefficient. This means full electromechanical coupling is retained. The theory also include the coupling between bending and extension and has been tested in various cases, for bending and sensing applications. Even if an analytical solution can be obtained in simplified cases, with the strong form of the equations, for most applications, a numerical method has to be used. In this thesis, a finite element method has been used to discretize the equation in space and a generalize alpha method has been used for time. This was the subject of chapter 4.

Handling contact with plate is much more difficult than in beam due to contact zone which is now a point, a curve or a surface. In addition, it has been explained why the strong form of the equations can not be used on the whole domain, which explains why, in the literature, contact problem for plates are solved using finite element methods. Reviewing paper for contact problems involving plates have not been encountered, so a comparison of frequently used methods and a classification of them has been given in chapter 5. In addition, a primal-dual active set strategy has been proposed, as a robust solution to tackle contact constraint in a finite element framework.

The proposed numerical scheme has been applied to both micropumps as benchmark cases and compared to 3D model made in Comsol Multiphysics software. A huge saving in computational time and memory requirement has been noted in sta-

tionary but also time dependant study. In addition, it has put forward the convergence issues that arises when the contact condition is simply modelled by a spring like force. Indeed, even if penetration in the obstacle can not be avoided, a high value for spring stiffness allows to reduce it, but a too high value generally leads to convergence issues, for the nonlinear solver in static studies which were performed. In addition, even if termination occurs without showing convergence issue, the solution obtained is sometimes far from the one obtained using the proposed scheme. A reduction of the spring stiffness seems to solve this problem and shows that there is a trade off here in the choice of the stiffness parameter, which has to be tuned according to the study, the applied load, the correspondent stiffness of the plate, the authorized penetration and maybe other parameters.

7.1 Original contributions

In this thesis, two original peristaltic and valveless micropumps have been proposed. If they are already a contribution in themselves, they have lead to several other contributions which can be summed up as follows:

- Development of an improved laminated beam theory: compared to equations usually used in literature, the electric field variation in the thickness of piezoelectric bending actuator and its impact on the stresses is now taken into account. In addition, loads due to Lorentz force are taken into account. This was presented in Beckers and Dehez (2013b) and Beckers and Dehez (2014a).
- Showing the limits of the usual assumptions made in simplified models of structure, i.e. null yy -stress or null yy -strain, in the case of piezoelectric bending structures and micropump, as explained in Beckers and Dehez (2013a).
- Development of a laminated plate theory, which takes into consideration bending-extension coupling, electric field variation across thickness of piezoelectric layers and loads due to Lorentz force, for which a first publication has been done in Beckers and Dehez (2014b). The unknowns are limited to the mechanical displacement while keeping full electromechanical coupling in piezoelectric materials.
- Development of a numerical scheme based on finite element for the aforementioned equations, with contact constraint handling, and its application to micropumps, as in Beckers and Dehez (2017).
- Classification of methods encountered in the literature for plate contact problems.

7.2 Outlook

If this thesis has established some milestones in the modelling of the proposed designs, one of the bigger restriction is the fluid, which is not taken into account in the models. The next step in the modelling process should be to add this one. However, this is a quite challenging problem, which has been investigated only superficially through literature, and from which several challenges can be extracted.

The first one is to choose correctly the set of equations to solve. Looking at the literature, fluid flows are usually modelled using Navier-Stokes equations either in compressible or incompressible regime. For flows in geometry with high aspect ratio, such as in the case of the micropumps considered here, these equations can generally be simplified, leading to equations such as the one for lubrication regime. A special field treating the problem at the interface between mechanics, contacting bodies, and flows, considering roughness, is known as elastohydrodynamics, for example in Zhu and Wang (2011), and is still a subject under research.

Once the set of equations is chosen, a proper method for solving these equations has to be found. Considering the micropumps, it was assumed that there was no fluid, or at least not the pumped one, between the diaphragm when they are in contact. The validity of this assumption is still an open question, but it adds some difficulties in the discretization using a mesh such as in the classical finite element method. Indeed, it means that at rest, there is fluid at the inlet and at the outlet only. When the diaphragm are actuated, a bubble of fluid is formed and then moved, until it is separated from the inlet. At the very moment where this separation occurs, a topological change in the fluid domain is performed, which is quite difficult to tackle with traditional techniques. A possibility is to use a fixed mesh approach with an immersed boundary technique, such as firstly introduced in Peskin (2002), or an immersed interface, such as in the explicit jump immersed interface method in Wiegmann and Bube (2000) or Le et al. (2006), or the cut-cell method in Pasquariello et al. (2016). Another technique, which has appeared quite recently, is to use the eXtended Finite Element Method, also known as XFEM, to capture the discontinuity directly through the shape functions. This method, already used in mechanics to capture discontinuity, such as in crack propagation problems, uses an enrichment of the element near the discontinuity with a proper integration method, such that the mesh has not to be conformal to the geometry. This method has already been applied to fluid structure interaction problems involving contact, such as in Mayer et al. (2010).

The discretization is not the only challenge when incompressible Navier-Stokes equations are used. Indeed, the incompressibility condition, i.e. the divergence free condition on the velocity field, is a constraint without any time-dependent equations to evolve it. Coping with this constraint is usually often done through a projection

method where a Poisson equation has to be solved. Proper conditions for this equation has to be used and pressure has to be fixed at one point in the domain, at least. But when a topological change occurs in the fluid domain, this pressure has also to be fixed inside the new domain. Since there is not any time dependence, fixing the pressure has to be done carefully, respecting the fluid-structure coupling. There is a lack in the literature in this kind of fluid-structure-contact problem and this question on how to fix the pressure properly is still open. A mean to avoid this problem could be to use a weakly compressible regime, with an equation of state such as the Tait equation, where the wave speed is not too high, to avoid too restrictive condition on the time step. Less classical methods such as the Lattice Boltzmann method could also be used since, for this method, no pressure equation has to be solved, instead pressure is reconstructed locally from distribution functions.

The question of the thermal deformations has also to be investigated. These could generate several problems such as thermal buckling, an instability that can make the micropump inoperative or put severe constraints on its sizing. Thus it is possible that they have to be taken into account in the model.

Another research direction could be to study numerically and experimentally both micropumps to validate the model. It would be the opportunity to verify that the tool considers enough of the physical phenomena.

In order to go further with the electromagnetic micropumps, several things could be studied. A first one could be to use the developed model to study different powering schemes and maybe adapting the design of the flexible printed circuit board to use well known switching power electronics circuits. Finding specific applications, in which the micropump could be deeply integrated could be interesting. As a such application is the one given in Homsy et al. (2007), where a magnetohydrodynamic micropump is used in a nuclear magnetic resonance system for chemical analysis. The field, which is necessary for the chemical analysis technique, can also be used as a replacement of the permanent magnets. In addition, in order to simplify the design, it seems that a non-symmetric version of the pump can greatly simplify the fluid management in and out of the pump, which is a tricky task.

Materials' data



This appendix regroups the data concerning the materials used for simulations made in this thesis. It is divided in two, between the elastic material and the piezoelectric material.

Linear elastic material For linear elastic materials the constitutive equations are given by Hooke's law. In this case and assuming isotropic material, the stresses and strains are linked by

$$\begin{bmatrix} S_{xx} \\ S_{yy} \\ S_{zz} \\ S_{yz} \\ S_{zx} \\ S_{xy} \end{bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1+\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 1+\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 1+\nu \end{bmatrix} \begin{bmatrix} T_{xx} \\ T_{yy} \\ T_{zz} \\ T_{yz} \\ T_{zx} \\ T_{xy} \end{bmatrix} \quad (\text{A.1})$$

where E is the Young's modulus and ν , the Poisson's ratio. The material used are copper, kapton, brass, Al 2014-T6 aluminum and AISI 4340 steel for which values of Young's modulus and Poisson ratio are given in table A.1.

Materials	E [GPa]	ν	ρ [kg/m ³]
Copper	124	0.33	8700
Kapton	2.5	0.34	1420
Brass	97	0.31	8490
Al 2014-T6 aluminum	72.4	0.33	2800
AISI 4340 steel	205	0.28	7850

Table A.1: Materials data

Piezoelectric material In the case of a layer made of piezoelectric material there is a coupling between the mechanical variables i.e. stress and strain fields and the electrical variables i.e. electrical and electrical displacement fields. In the case of an orthotropic material polarized in the z direction such as piezoelectric ceramics, the strain-charge form is given by:

$$\begin{bmatrix} S_{xx} \\ S_{yy} \\ S_{zz} \\ S_{yz} \\ S_{zx} \\ S_{xy} \end{bmatrix} = \begin{bmatrix} s_{11}^E & s_{12}^E & s_{13}^E & 0 & 0 & 0 \\ s_{21}^E & s_{22}^E & s_{23}^E & 0 & 0 & 0 \\ s_{31}^E & s_{32}^E & s_{33}^E & 0 & 0 & 0 \\ 0 & 0 & 0 & s_{44}^E & 0 & 0 \\ 0 & 0 & 0 & 0 & s_{55}^E & 0 \\ 0 & 0 & 0 & 0 & 0 & s_{66}^E \end{bmatrix} \begin{bmatrix} T_{xx} \\ T_{yy} \\ T_{zz} \\ T_{yz} \\ T_{zx} \\ T_{xy} \end{bmatrix} + \begin{bmatrix} 0 & 0 & d_{31} \\ 0 & 0 & d_{32} \\ 0 & 0 & d_{33} \\ 0 & d_{24} & 0 \\ d_{15} & 0 & 0 \end{bmatrix} \begin{bmatrix} E_1 \\ E_2 \\ E_3 \end{bmatrix} \quad (\text{A.2})$$

$$\begin{bmatrix} D_1 \\ D_2 \\ D_3 \end{bmatrix} = \begin{bmatrix} 0 & 0 & 0 & 0 & d_{15} & 0 \\ 0 & 0 & 0 & d_{24} & 0 & 0 \\ d_{31} & d_{32} & d_{33} & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} T_{xx} \\ T_{yy} \\ T_{zz} \\ T_{yz} \\ T_{zx} \\ T_{xy} \end{bmatrix} + \begin{bmatrix} \epsilon_{11} & 0 & 0 \\ 0 & \epsilon_{22} & 0 \\ 0 & 0 & \epsilon_{33} \end{bmatrix} \begin{bmatrix} E_1 \\ E_2 \\ E_3 \end{bmatrix} \quad (\text{A.3})$$

where the s_{ij}^E is the compliance coefficient under constant electric field, d_{31} is the piezoelectric coupling coefficient. The piezoelectric material used in this thesis is the PZT-5H, for which the mass density is 7500 km, and the previously defined matrix are given by:

$$s^E = \begin{bmatrix} 16.5 & -4.78 & -8.45 & 0 & 0 & 0 \\ & 16.5 & -8.45 & 0 & 0 & 0 \\ & & 20.7 & 0 & 0 & 0 \\ & & & 43.5 & 0 & 0 \\ & & & & 43.5 & 0 \\ & & & & & 42.6 \end{bmatrix} \times 10^{-12} [\text{Pa}^{-1}] \quad (\text{A.4})$$

$$d = \begin{bmatrix} 0 & 0 & 0 & 0 & 7.41 & 0 \\ 0 & 0 & 0 & 7.41 & 0 & 0 \\ -2.74 & -2.74 & 5.93 & 0 & 0 & 0 \end{bmatrix} \times 10^{-10} [\text{C N}^{-1}] \quad (\text{A.5})$$

$$\epsilon = \begin{bmatrix} 3130 & 0 & 0 \\ & 3130 & 0 \\ & & 3400 \end{bmatrix} \quad (\text{A.6})$$

Another possible form of those constitutive equations are given by the stress-charge form:

$$\begin{bmatrix} T_{xx} \\ T_{yy} \\ T_{zz} \\ T_{yz} \\ T_{zx} \\ T_{xy} \end{bmatrix} = \begin{bmatrix} c_{11}^E & c_{12}^E & c_{13}^E & 0 & 0 & 0 \\ c_{21}^E & c_{22}^E & c_{23}^E & 0 & 0 & 0 \\ c_{31}^E & c_{32}^E & c_{33}^E & 0 & 0 & 0 \\ 0 & 0 & 0 & c_{44}^E & 0 & 0 \\ 0 & 0 & 0 & 0 & c_{55}^E & 0 \\ 0 & 0 & 0 & 0 & 0 & c_{66}^E \end{bmatrix} \begin{bmatrix} S_{xx} \\ S_{yy} \\ S_{zz} \\ S_{yz} \\ S_{zx} \\ S_{xy} \end{bmatrix} - \begin{bmatrix} 0 & 0 & e_{31} \\ 0 & 0 & e_{32} \\ 0 & 0 & e_{33} \\ 0 & e_{24} & 0 \\ e_{15} & 0 & 0 \end{bmatrix} \begin{bmatrix} E_1 \\ E_2 \\ E_3 \end{bmatrix} \quad (\text{A.7})$$

$$\begin{bmatrix} D_1 \\ D_2 \\ D_3 \end{bmatrix} = \begin{bmatrix} 0 & 0 & 0 & 0 & e_{15} & 0 \\ 0 & 0 & 0 & e_{24} & 0 & 0 \\ e_{31} & e_{32} & e_{33} & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} S_{xx} \\ S_{yy} \\ S_{zz} \\ S_{yz} \\ S_{zx} \\ S_{xy} \end{bmatrix} + \begin{bmatrix} \epsilon_{11}^S & 0 & 0 \\ 0 & \epsilon_{22}^S & 0 \\ 0 & 0 & \epsilon_{33}^S \end{bmatrix} \begin{bmatrix} E_1 \\ E_2 \\ E_3 \end{bmatrix}. \quad (\text{A.8})$$

where the matrices are given by:

$$C^E = \begin{bmatrix} 127.205 & 80.2122 & 84.6702 & 0 & 0 & 0 \\ & 127.205 & 84.6702 & 0 & 0 & 0 \\ & & 117.4360 & 0 & 0 & 0 \\ & & & 22.9885 & 0 & 0 \\ & & & & 22.9885 & 0 \\ & & & & & 23.4742 \end{bmatrix} [\text{GPa}] \quad (\text{A.9})$$

$$e = \begin{bmatrix} 0 & 0 & 0 & 0 & 17.0345 & 0 \\ 0 & 0 & 0 & 17.0345 & 0 & 0 \\ -6.62281 & -6.62281 & 23.2403 & 0 & 0 & 0 \end{bmatrix} [\text{C m}^{-2}] \quad (\text{A.10})$$

$$\epsilon^S = \begin{bmatrix} 1704.4 & 0 & 0 \\ & 1704.4 & 0 \\ & & 1433.6 \end{bmatrix} \quad (\text{A.11})$$

Finite element - technical and computational aspects

B

Among the various numerical methods which are developed in the literature, the finite element is one of the most popular one, with several commercial software available. This is explained by its ability to solve many kind of partial differential equation on complex geometrical domains. In addition, its modularity allows to add additional physics or elements to a software without rewriting everything. An other advantage is its strong mathematical basement, which allows to find bounds on error, for example, using functional analysis.

The aim of this appendix is not to give a deep insight of all the possibilities given by the method, but rather to explain some of the basics, as well as giving some tips to help in writing its own codes and understanding the ones developed in this thesis. In addition, it will give the expression of the various matrices of the laminated plate equations developed in chapter 4. Readers who want additional information to jump in the finite element method, in a practical way, can referred to Whiteley (2017).

In the following, the finite element method will be explained from the domain discretization to the post processing step. To support the text, some short Matlab scripts are provided to show how the method is implemented. Instead of giving the code for the laminated plate equations, coupling extensional and bending, it is preferred here to show the basics on a simpler problem, the bending of the middle plane of a rectangular plate, made of only one layer, clamped on its four sides and loaded by a constant pressure. This problem allows to show the helpful tricks to implement the whole problem. In addition, the details of the mass and stiffness matrices, as well as of the load vector, for the laminated plate theory, developed in chapter 4, are given. here.

Before starting, let reminds of the weak equation for the simplified model considered here. The plates is clamped on its four sides which means the displacement

w , as well as its first normal derivative $\nabla w \cdot \hat{n}$, are zero on the edges. As developed in Fernandes et al. (2001), the equation to solve is thus:

$$\frac{Et^3}{12(1-\nu^2)} \int_{\Omega} \tilde{\mathbf{S}}^t \mathbf{D} \mathbf{S} d\Omega = \int_{\Omega} f \tilde{w} d\Omega \quad (\text{B.1})$$

where $\mathbf{S}$ and $\tilde{\mathbf{S}}$ are respectively, the strain of the plate and the virtual strain, $\tilde{w}$, the virtual displacement, E , the Young's modulus, ν , the Poission ratio, t , the plate's thickness and f , the applied pressure, $\mathbf{S}$ and $\mathbf{D}$ are given by:

$$\mathbf{S} = \begin{pmatrix} w_{,xx} \\ w_{,yy} \\ w_{,xy} \end{pmatrix} \quad (\text{B.2})$$

$$\mathbf{D} = \begin{pmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{pmatrix} \quad (\text{B.3})$$

B.1 The mesh

The finite element method is based on the decomposition of the computational domain in simple geometrical shapes, such as triangles or quadrilaterals for 2D domains, as shown in fig. B.1. Each shape is described by vertices and edges and form what is called a mesh. Informations about the mesh are provided in a connectivity matrix, where line i gives the vertices corresponding to element i . The connectivity matrix associated to the mesh shown in fig. B.1 is given by:

$$\text{elem2nodes} = \begin{bmatrix} 1 & 2 & N+3 & N+2 \\ 2 & 3 & N+4 & N+3 \\ \vdots & \vdots & \vdots & \vdots \end{bmatrix} \quad (\text{B.4})$$

An other matrix, where line i gives the coordinates of the vertex i , is also used to describe the geometry. The mesh can be made by the user or automatically using a meshing routine. The quality of the mesh has a big influence over the solution. A poor mesh can lead to convergence problem and reduce the solution accuracy. Making a good mesh can be a hard job which ask to iterate over it. It should be fine enough in zones with high gradients to capture the solution properly and coarse enough in zone where the solution varies less to limit computational costs. In addition the aspect ratio of the shape is also important and is taken into account in measure of mesh quality. A simple mesh generator, for mesh made of triangles, as well as some

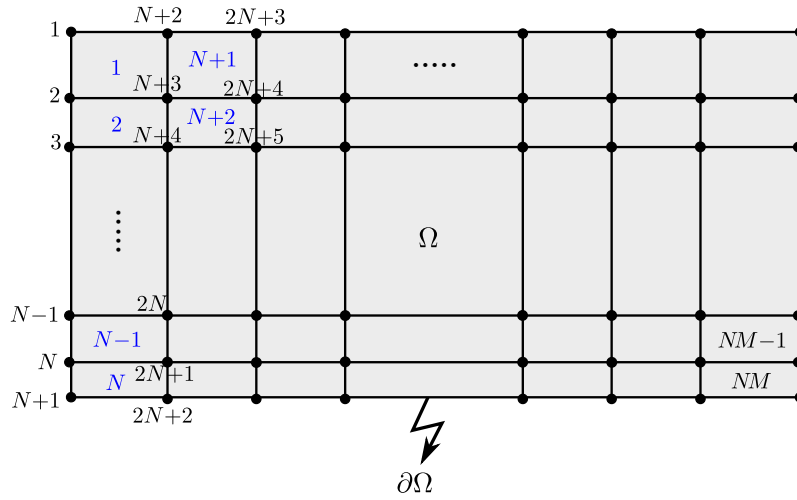


Fig. B.1: Plate geometry and its associated mesh, made of rectangles. At each vertex and rectangle is associated a number, which will be reported in a matrix, called the connectivity matrix, to make the connection between each shape and each vertex.

informations over mesh quality is provided in Persson and Strang (2004). In this thesis, a mesh made of rectangle is used. A small script, similar as the one used for the laminated plate code, showing the definition of the mesh is provided in the script B.1.

Matlab Script B.1: Generating the mesh and the connectivity matrix, amongst others.

```

1 function mesh = generateMesh(plate,elemLMax,minSub)
2
3 sizeX = plate.length;
4 sizeY = plate.height;
5 Nx = max(ceil(sizeX./elemLMax),minSub);
6 Ny = max(ceil(sizeY./elemLMax),minSub);
7 Nelem = Nx*Ny;
8
9 x = linspace(0,sizeX,Nx+1);
10 y = linspace(0,sizeY,Ny+1);
11
12 [X,Y] = meshgrid(x,y);
13 Nnodes = numel(X);
14 nodesNumbering = reshape(1:Nnodes,size(X));
15
16 % Connectivity matrix
17 elem2nodes = [reshape(nodesNumbering(2:end,1:end-1),[],1),...
18               reshape(nodesNumbering(2:end,2:end),[],1),...
19               reshape(nodesNumbering(1:end-1,2:end),[],1),...
```

```

20         reshape(nodesNumbering(1:end-1,1:end-1),[],1)];
21
22 % Boundary conditions
23 clampedBCx = [nodesNumbering(:,1); nodesNumbering(:,end)];
24 clampedBCy = [reshape(nodesNumbering(1,:),[],1);...
25               reshape(nodesNumbering(end,:),[],1)];
26 h = [sizeX/Nx,sizeY/Ny];
27
28 mesh.FixedW = [clampedBCx; clampedBCy];
29 mesh.FixedWx = clampedBCx+Nnodes;
30 mesh.FixedWy = clampedBCy+2*Nnodes;
31
32
33 % Nodes where the equation has to be solved
34 FreeNodes = 1:4*Nnodes;
35 FreeNodes([mesh.FixedW;mesh.FixedWx;mesh.FixedWy]) = [];
36
37 % Formatting the output
38 mesh.Nx = Nx;
39 mesh.Ny = Ny;
40 mesh.Nnodes = Nnodes;
41 mesh.Nelem = Nelem;
42 mesh.gridX = x;
43 mesh.gridY = y;
44 mesh.X = X;
45 mesh.Y = Y;
46 mesh.h = h;
47 mesh.elem2nodes = elem2nodes;
48 mesh.clampedBCx = clampedBCx;
49 mesh.clampedBCy = clampedBCy;
50 mesh.FreeNodes = FreeNodes;
51 end

```

B.2 The element

Over each of this shape an approximation for the solution is built using polynomials $\phi_i(x)$ with compact support i.e. they are non zero only on the shape and zero over the rest of the domain. Those functions interpolate the value of the solution given at the nodes. Those nodes can be the vertices of the shape but are not restrained to them. The polynomials used for interpolation have the following property:

$$\phi_i(x_j) = \begin{cases} 1 & \text{if } i = j \\ 0 & \text{if } i \neq j \end{cases} \quad (\text{B.5})$$

The shape, the nodes, the polynomials as well as the discrete values defined at each nodes form together what is called an element. Various elements have been developed in the literature, with different properties conferred by the polynomials,

as well as the choices of nodes and discretized variables. One which is important is its continuity across its edges. When working with conformal finite element, the continuity of the function up to $m - 1$ derivatives where m is the higher derivative's degree has to be ensured. For example, in the plate equation the higher degree is 2 which means the element has to be C^1 . Non conforming finite element are also possible but will not be discussed further here.

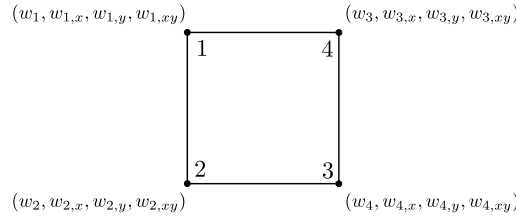


Fig. B.2: Bogner-Fox-Schmit element. Each vertex of the rectangle is also a node, where the discretized value used for interpolation are the one of w , w_x , w_y and w_{xy} . Shape functions used for interpolation are tensor products of one dimensional cubic Hermite polynomials.

In this thesis, and since the geometry is simple, a rectangular element called Bogner-Fox-Schmit (BFS), which is shown in fig. B.2, has been chosen and enhanced. BFS element has been used in Fernandes et al. (2001) to study plate bending in case of obstacle and has shown good results. It has however a strong restriction since its edges have to remain parallel. An other well known triangular element that can be used to discretized more complex domain is the Argyris element.

In this thesis, three functions have to be discretized u_0 , v_0 and w . Since u_0 and v_0 only require a C^0 interpolation, they are both discretized using bilinear functions which are just tensor product of one dimensional linear function. The approximate displacement u_0^h and v_0^h in the reference element are then given by

$$u_h(\xi, \eta) = \Phi^T \mathbf{U} \quad (\text{B.6})$$

$$v_h(\xi, \eta) = \Phi^T \mathbf{V} \quad (\text{B.7})$$

where

$$\Phi = \frac{1}{4} [(1 - \xi)(1 - \eta), (1 + \xi)(1 - \eta), \dots, (1 + \xi)(1 + \eta), (1 - \xi)(1 + \eta)]^T. \quad (\text{B.8})$$

The vectors $\mathbf{U}$ and $\mathbf{V}$ contain the discrete values for u_0 and v_0 respectively. w is discretized using bicubic Hermite polynomials. The approximation w_h of the displacement w is given by

$$w_h(\xi, \eta) = \Psi_{W_1}^T \mathbf{W}_1 + \frac{\delta x}{2} \Psi_{W_2}^T \mathbf{W}_2 + \frac{\delta y}{2} \Psi_{W_3}^T \mathbf{W}_3 + \frac{\delta x \delta y}{4} \Psi_{W_4}^T \mathbf{W}_4 \quad (\text{B.9})$$

where $\mathbf{W}_1, \mathbf{W}_2, \mathbf{W}_3$ and $\mathbf{W}_4$ are vectors with the four nodal values for w, w_x, w_y, w_{xy} respectively. $\Psi_{W_1}^T, \Psi_{W_2}^T, \Psi_{W_3}^T$ and $\Psi_{W_4}^T$ are vectors of associated shape functions. Those are tensor products of cubic Hermite polynomials $H(s), H'(s)$

$$\begin{aligned} \Psi_{W_1} &= [H_{-1}(\xi)H_{-1}(\eta), H_1(\xi)H_{-1}(\eta), \dots \\ &\quad H_1(\xi)H_1(\eta), H_{-1}(\xi)H_1(\eta)]^T \\ \Psi_{W_2} &= [H'_{-1}(\xi)H_{-1}(\eta), H'_1(\xi)H_{-1}(\eta), \dots \\ &\quad H'_1(\xi)H_1(\eta), H'_{-1}(\xi)H_1(\eta)]^T \\ \Psi_{W_3} &= [H_{-1}(\xi)H'_{-1}(\eta), H_1(\xi)H'_{-1}(\eta), \dots \\ &\quad H_1(\xi)H'_1(\eta), H_{-1}(\xi)H'_1(\eta)]^T \\ \Psi_{W_4} &= [H'_{-1}(\xi)H'_{-1}(\eta), H'_1(\xi)H'_{-1}(\eta), \dots \\ &\quad H'_1(\xi)H'_1(\eta), H'_{-1}(\xi)H'_1(\eta)]^T \end{aligned} \quad (\text{B.10})$$

where

$$\begin{aligned} H_{-1}(s) &= \frac{1}{4}(s-1)^2(s+2) \\ H_1(s) &= \frac{1}{4}(s+1)^2(2-s) \\ H'_{-1}(s) &= \frac{1}{4}(s-1)^2(s+1) \\ H'_1(s) &= \frac{1}{4}(s+1)^2(s-1) \end{aligned} \quad (\text{B.11})$$

are one dimensional cubic Hermite polynomials, which are non zero only in the element. In addition those polynomials take the value one at only one node for either the polynomial or its first derivative. Evaluated at the other nodes their value is zero. w_h is compactly written as

$$w_h(\xi, \eta) = \Psi^T \mathbf{W} \quad (\text{B.12})$$

where $\mathbf{W} = [\mathbf{W}_1, \mathbf{W}_2, \mathbf{W}_3, \mathbf{W}_4]^T$ and Ψ is the according shape functions vector. Approximations (4.72), (4.73) and (4.74) can be used in the weak equations restricted to the element domain Ω^e . The four vertices of the rectangle are the nodes and at each of them the discrete value of u_0, v_0, w, w_x, w_y and w_{xy} are stored.

For the study of the simple plate equation, used in this appendix to explain the implementation, the classic BFS element, as shown in fig. B.2, is used. A plot of the shape functions is shown in fig. B.3, on the reference element $\hat{\Omega} = [-1, 1]^2$.

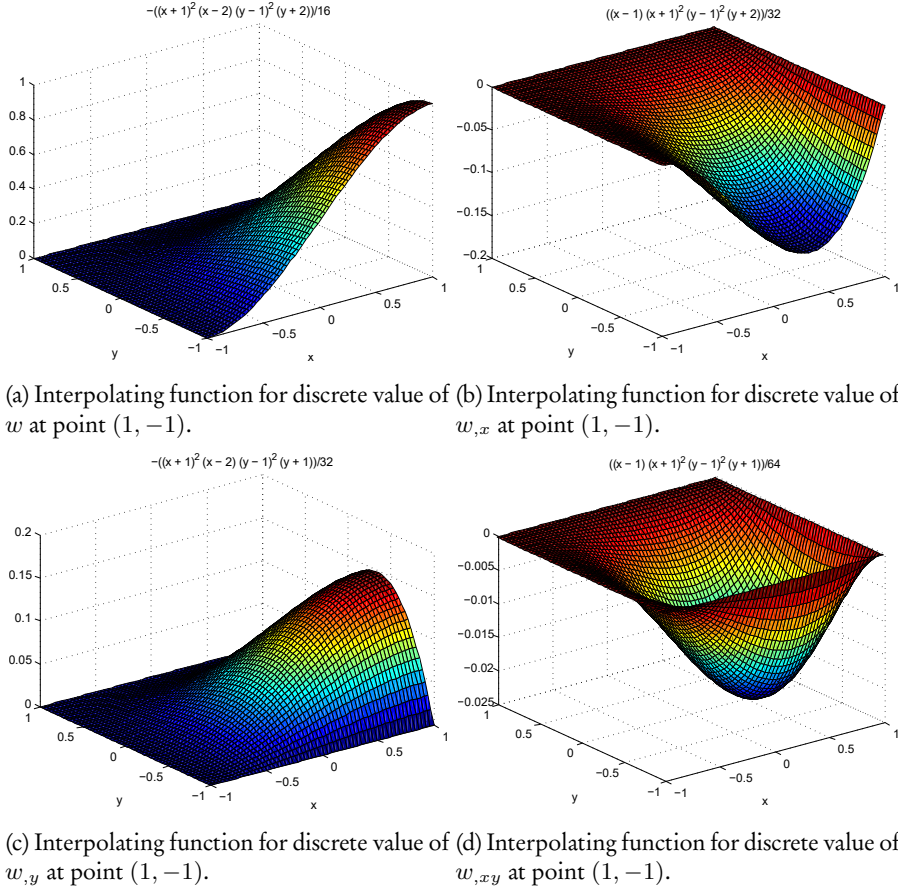


Fig. B.3: Bicubic Hermite polynomials used as shape functions in Bogner-Fox-Schmit element, for the node $(1, -1)$, in the reference domain. At other nodes that $(1, -1)$, the functions are null. In addition, at point $(1, -1)$, each function takes a value of 1 only for the discrete value to which it corresponds and 0 for others.

B.3 Local contribution and matrix assembly

Using a Galerkin method, the virtual displacements are discretized using the same basis functions as for the displacement. The weak form is then applied to solve the equation on the whole domain. Since the shape functions have compact support, i.e. they are non zero only on the element where they are defined, the weak form can be applied element by element, with boundary terms only applied on the external boundary $\partial\Omega$ when necessary, i.e. when the displacement is not fixed. To compute the integrals, most of finite element codes mapped the current element to a reference one, using a mapping function, and then perform numerical integration. The Jacobian of the transformation is used to transform the integral from the element to the reference one. In this thesis, since the element is quite simple, the integrals can

be computed analytically and then evaluated. This has been done using the Matlab symbolic capabilities to define the shape function on the reference element, compute their derivatives, computing each integrals for the mass and stiffness matrix, as well as for the load vector, and then to create a function to evaluate them numerically. The code, which is shown in B.2 for the simplified model, has to be run once to create the function and then, it can be used to evaluate the local contribution of each element. This helps to reduce the computational cost for matrix assembly.

Matlab Script B.2: Symbolic computation of stiffness matrix and load vector for the simplified plate problem.

```

1  syms x y dx dy nu E t
2  assume(x,'real');
3  assume(y,'real');
4  assume(dx,'real');
5  assume(dy,'real');
6
7  %%% Stiffness matrix %%%
8
9  % cubic Hermite polynomials with respect to x
10 Nx11 = ((2+x)*(1-x)^2)/4;
11 Nx12 = ((2-x)*(1+x)^2)/4;
12 Nx21 = ((1+x)*(1-x)^2)*dx/8;
13 Nx22 = -((1-x)*(1+x)^2)*dx/8;
14
15 % cubic Hermite polynomials with respect to y
16 Ny11 = ((2+y)*(1-y)^2)/4;
17 Ny12 = ((2-y)*(1+y)^2)/4;
18 Ny21 = ((1+y)*(1-y)^2)*dy/8;
19 Ny22 = -((1-y)*(1+y)^2)*dy/8;
20
21 % Shape function for Bogner-Fox-Schmit element
22 D0 = [Nx11*Ny11 Nx12*Ny11 Nx12*Ny12 Nx11*Ny12]; % Interpolate w
23 Dx = [Nx21*Ny11 Nx22*Ny11 Nx22*Ny12 Nx21*Ny12]; % Interpolate ...
    dw/dx
24 Dy = [Nx11*Ny21 Nx12*Ny21 Nx12*Ny22 Nx11*Ny22]; % Interpolate ...
    dw/dy
25 Dxy = [Nx21*Ny21 Nx22*Ny21 Nx22*Ny22 Nx21*Ny22]; % Interpolate ...
    d²w/dxdy
26
27 W = [D0 Dx Dy Dxy];
28
29 curvXX = simplify(diff(diff(W,x),x)*(2/dx)^2);
30 curvYY = simplify(diff(diff(W,y),y)*(2/dy)^2);
31 curvXY = simplify(diff(diff(W,x),y)*(2/dx)*(2/dy));
32
33 S = [curvXX ; curvYY ; 2*curvXY];
34 D = ((t^3)*E/(12*(1-nu^2)))*[1 nu 0; nu 1 0; 0 0 (1-nu)/2];
35
36 K = int(int(S'*D*S,x,-1,1),y,-1,1)*dx*dy/4; % stiffness matrix

```

```

37 fpressure = int(int(W',x,-1,1)*dx/2,y,-1,1)*(dy/2); % load vector
38
39 % Generating matlab function for numerical evaluation
40 matlabFunction(K,'File','elemStiffMat');
41 matlabFunction(fpressure,'File','elemPressureLoad');

```

In this thesis and in the simplified problem studied in this appendix, the rectangular element is mapped to the reference one, which is a square with domain $\hat{\Omega} = [-1, 1]^2$, using the transformation:

$$F : (x, y) \rightarrow (\xi, \eta) := \left(\frac{2}{\delta x}(x - x_c), \frac{2}{\delta y}(y - y_c) \right) \quad (\text{B.13})$$

where (x_c, y_c) are the coordinates of the element's center and $\delta x, \delta y$ are the dimensions in x and y directions. Since each element is defined by the interpolation of 6 discrete values on 4 nodes in the laminated plate equation, each local mass and stiffness matrices are 24 by 24. For the simple plate equation, only 4 discrete values per nodes have to be interpolated, leading to 16 by 16 local matrices. Those local contributions have to be assembled to form the matrices of the system, taking into consideration that adjacent elements share discrete values. In order to construct those matrices, the connectivity matrix is used to place the value at the correct position in the matrix. The process is shown in fig. B.4, for the simple plate model.

Matlab Script B.3: Matrix assembly.

```

1 function [K,Fp] = Assemble(plate,mesh)
2
3 Nnodes = mesh.Nnodes;
4 Nelem = mesh.Nelem;
5 elem2nodes = mesh.elem2nodes;
6 h = mesh.h;
7 E = plate.E;
8 nu = plate.nu;
9 t = plate.thickness;
10
11 [Krow,Kcol,Kval] = deal(zeros(Nelem,16*16));
12 [frow,fcol,fp] = deal(zeros(Nelem,16));
13
14 for i=1:mesh.Nelem
15     nodes = elem2nodes(i,:);
16     map = [nodes, nodes+Nnodes, nodes+2*Nnodes, nodes+3*Nnodes];
17
18     Krow(i,:) = kron(ones(1,16),map);
19     Kcol(i,:) = kron(map,ones(1,16));
20     Kval(i,:) = reshape(elemStiffMat(E,h(1),h(2),nu,t),1,[]);
21
22     frow(i,:) = map;

```

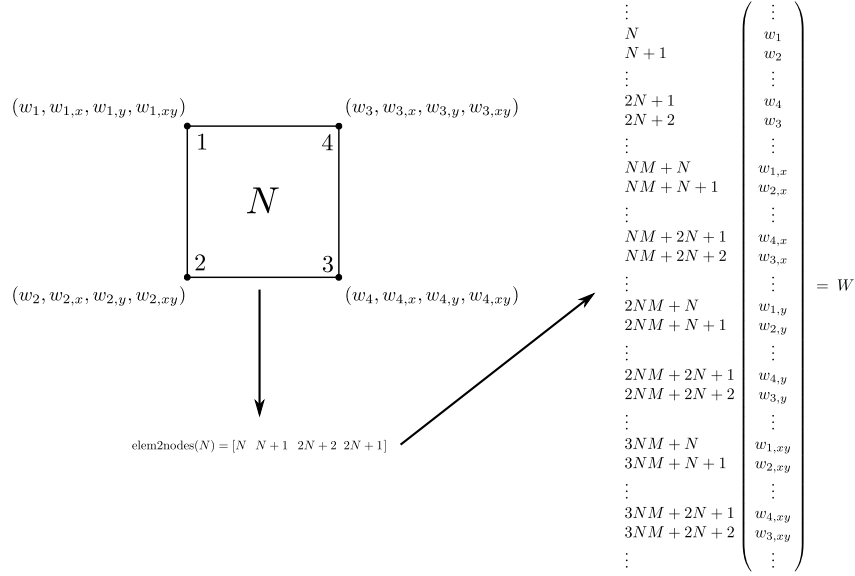


Fig. B.4: Matrix assembly process. The connectivity matrix is used to construct the global matrices, placing the local contribution at the right place with respect to the global numbering used. The corresponding Matlab code is given in B.3.

```

23     fcol(i,:) = i*ones(1,16);
24     fp(i,:) = reshape(elemPressureLoad(h(1),h(2)),1,[]);
25 end
26
27 K = sparse(Krow,Kcol,Kval,4*Nnodes,4*Nnodes);
28 Fp = sparse(frow,fcol,fp,4*Nnodes,Nelem);
29 end

```

This can be done quite easily using the ‘sparse’ function in Matlab, as shown in the script B.2 for the stiffness matrix and load vector of the simplified model. Once the matrices and the load vector are obtained, they can be used to obtain the effective stiffness matrix, $\mathbf{K}_{eff}$ and load vector, $\mathbf{g}_n$ and then solved to obtain the incremental displacement. This is done using the backslash operator in Matlab, as shown in the script B.4 where the deflection is computed.

Matlab Script B.4: Computation of a square plate loaded by a constant pressure

```

1 plate.length = 40e-3;
2 plate.height = 40e-3;
3 plate.E = 205e9;
4 plate.nu = 0.28;
5 plate.thickness = 1e-3;
6

```

```

7  elemLMax = 1e-3;
8  minSub = 10; % Minimum number of subdivision in one dimension
9
10 mesh = generateMesh(plate,elemLMax,minSub);
11 pressure = -1e7*ones(mesh.Nelem,1);
12
13 [K,Fp] = Assemble(plate,mesh);
14 free = mesh.FreeNodes;
15 fp = Fp*pressure;
16
17 W = zeros(4*mesh.Nnodes,1);
18 W(free) = K(free,free)\fp(free);
19 w = reshape(W(1:mesh.Nnodes),size(mesh.X));
20
21 surf(mesh.X,mesh.Y,w)

```

B.4 Post-processing

Once the system is solved, discrete values for the displacement, and for the model of the dynamics of the laminated plates, its first and second time derivatives, are known. Additional quantities of interest such as the electric field or the stress can be computed from it. In order to do that, matrices evaluating the necessary values, such as second spatial derivatives of w and first spatial derivative of u_0 and v_0 , at given points are assembled in the same fashion as for mass and stiffness matrix. Evaluation of the stress is usually performed at Gauss points, i.e. the points used to compute the integrals, but here no such points are used. In this thesis, if not stated otherwise, they are evaluated at the element's center. If it is technically possible to evaluate the stress, or any other quantity, at any desired point, it has to be noted that evaluating them on the edge on an element can lead to some issues. Indeed, the element used to discretize the equation ensures C^1 continuity, meaning that the second spatial derivative is not necessarily constant across an edge between two elements.

B.5 Laminated plates

The integrals to perform for the laminated plate equations developed in chapter 4 are reported here. The integrals are written on the reference element, already taking care of the transformation from the current element to the reference one, using the mapping function given earlier in this appendix in (B.13). The matrices appearing in 4.80 are given explicitly for a displacement vector $\mathbf{d} = [\mathbf{U}_0, \mathbf{V}_0, \mathbf{W}]^T$. The mass matrix $\mathbf{M}$ is given by:

$$\begin{pmatrix} -\mathbf{M}_{uu} & 0 & \mathbf{M}_{uw} \\ 0 & -\mathbf{M}_{vv} & \mathbf{M}_{vw} \\ \mathbf{M}_{uw}^T & \mathbf{M}_{vw}^T & -\mathbf{M}_{ww} \end{pmatrix} \quad (\text{B.14})$$

which shows clearly the couplings between each variables. Each submatrices are:

$$\mathbf{M}_{uu} = \int_{\hat{\Omega}} I^1 \Phi \Phi^T \frac{\delta x \delta y}{4} d\Omega \quad (\text{B.15})$$

$$\mathbf{M}_{vv} = \int_{\hat{\Omega}} I^1 \Phi \Phi^T \frac{\delta x \delta y}{4} d\Omega \quad (\text{B.16})$$

$$\mathbf{M}_{ww} = \int_{\hat{\Omega}} I^1 \frac{\delta x \delta y}{4} \Psi \Psi^T + I^3 \frac{\delta y}{\delta x} \Psi_{,\xi} \Psi_{,\xi}^T + I^3 \frac{\delta x}{\delta y} \Psi_{,\eta} \Psi_{,\eta}^T d\Omega \quad (\text{B.17})$$

$$\mathbf{M}_{uw} = \int_{\hat{\Omega}} I^2 \frac{\delta y}{2} \Phi \Psi_{,\xi}^T d\Omega \quad (\text{B.18})$$

$$\mathbf{M}_{vw} = \int_{\hat{\Omega}} I^2 \frac{\delta x}{2} \Phi \Psi_{,\eta}^T d\Omega. \quad (\text{B.19})$$

The stiffness matrix $\mathbf{K}$ is:

$$\begin{pmatrix} -\mathbf{K}_{uu} & -\mathbf{K}_{uv} & \mathbf{K}_{uw} \\ -\mathbf{K}_{uv}^T & -\mathbf{K}_{vv} & \mathbf{K}_{vw} \\ \mathbf{K}_{uw}^T & \mathbf{K}_{vw}^T & -\mathbf{K}_{ww} \end{pmatrix} \quad (\text{B.20})$$

where submatrices are:

$$\mathbf{K}_{uu} = \int_{\hat{\Omega}} \frac{\delta y}{\delta x} A_{11}^1 \Phi_{,\xi} \Phi_{,\xi}^T + \frac{\delta x}{\delta y} \frac{A_{66}^1}{2} \Phi_{,\eta} \Phi_{,\eta}^T d\Omega \quad (\text{B.21})$$

$$\mathbf{K}_{vv} = \int_{\hat{\Omega}} \frac{\delta x}{\delta y} A_{11}^1 \Phi_{,\eta} \Phi_{,\eta}^T + \frac{\delta y}{\delta x} \frac{A_{66}^1}{2} \Phi_{,\xi} \Phi_{,\xi}^T d\Omega \quad (\text{B.22})$$

$$\mathbf{K}_{uv} = \int_{\hat{\Omega}} A_{12}^1 \Phi_{,\xi} \Phi_{,\eta}^T + \frac{A_{66}^1}{2} \Phi_{,\eta} \Phi_{,\xi}^T d\Omega \quad (\text{B.23})$$

$$\mathbf{K}_{uw} = \int_{\hat{\Omega}} \frac{2}{\delta y} A_{12}^2 \Phi_{,\xi} \Psi_{,\eta\eta}^T + \frac{2}{\delta y} A_{66}^2 \Phi_{,\eta} \Psi_{,\xi\eta}^T + \frac{2\delta y}{\delta x^2} A_{11}^2 \Phi_{,\xi} \Psi_{,\xi\xi}^T d\Omega \quad (\text{B.24})$$

$$\mathbf{K}_{vw} = \int_{\hat{\Omega}} \frac{2}{\delta x} A_{12}^2 \Phi_{,\eta} \Psi_{,\xi\xi}^T + \frac{2}{\delta x} A_{66}^2 \Phi_{,\xi} \Psi_{,\xi\eta}^T + \frac{2\delta x}{\delta y^2} A_{11}^2 \Phi_{,\eta} \Psi_{,\eta\eta}^T d\Omega \quad (\text{B.25})$$

$$\begin{aligned} \mathbf{K}_{ww} = & \int_{\hat{\Omega}} \frac{4\delta y}{\delta x^3} A_{11}^3 \Psi_{,\xi\xi} \Psi_{,\xi\xi}^T + \frac{4}{\delta x \delta y} A_{12}^3 \Psi_{,\xi\xi} \Psi_{,\eta\eta}^T \\ & + \frac{4}{\delta x \delta y} A_{12}^3 \Psi_{,\eta\eta} \Psi_{,\xi\xi}^T + \frac{8}{\delta x \delta y} A_{66}^3 \Psi_{,\xi\eta} \Psi_{,\eta\xi}^T + \frac{4\delta x}{\delta y^3} A_{11}^3 \Psi_{,\eta\eta} \Psi_{,\eta\eta}^T d\Omega. \end{aligned} \quad (\text{B.26})$$

And finally the load vector is:

$$\begin{pmatrix} \mathbf{F}_u \\ \mathbf{F}_v \\ \mathbf{F}_w \end{pmatrix} \quad (\text{B.27})$$

with:

$$\mathbf{F}_u = \int_{\hat{\Omega}} \Phi_{,\xi} \gamma_0 \frac{\delta y}{2} d\Omega \quad (\text{B.28})$$

$$\mathbf{F}_v = \int_{\hat{\Omega}} \Phi_{,\eta} \gamma_0 \frac{\delta x}{2} d\Omega \quad (\text{B.29})$$

$$\mathbf{F}_w = \int_{\hat{\Omega}} -\Psi p \frac{\delta x \delta y}{4} + \gamma_1 \left(\Psi_{,\xi\xi} \frac{\delta y}{\delta x} + \Psi_{,\eta\eta} \frac{\delta x}{\delta y} \right) d\Omega. \quad (\text{B.30})$$

In all these equations, I^k and A_{ij}^k are taken from (4.11) and (4.44) respectively. In order to compute $\mathbf{F}_w$ using symbolic tools, the pressure is assumed to be constant on the element and is evaluated at its center.

Bibliography

- Ala'aldeen, T., Rawashdeh, N. A., Sanna, M., Büttgenbach, S., and Dietzel, A. (2015). Development of a novel electromagnetic double action meso-scale pump. In *Research and Education in Mechatronics (REM), 2015 16th International Conference on*, pages 250–254. IEEE.
- Amirouche, F., Zhou, Y., and Johnson, T. (2009). Current micropump technologies and their biomedical applications. *Microsystem technologies*, 15(5):647–666.
- Auricchio, F. and Sacco, E. (1996). Augmented lagrangian finite-elements for plate contact problems. *International journal for numerical methods in engineering*, 39(24):4141–4158.
- Beckers, G. and Dehez, B. (2013a). Design and modeling of a quasi-static peristaltic piezoelectric micropump. In *Electrical Machines and Systems (ICEMS), 2013 International Conference on*, pages 1301–1306. IEEE.
- Beckers, G. and Dehez, B. (2013b). Modelling of electric field and stress in piezoelectric composite under bending load in quasi-static conditions. In *Ultrasonics Symposium (IUS), 2013 IEEE International*, pages 2191–2194. IEEE.
- Beckers, G. and Dehez, B. (2014a). Design and modeling of an electromagnetic peristaltic micropump. In *Advanced Intelligent Mechatronics (AIM), 2014 IEEE/ASME International Conference on*, pages 180–185. IEEE.
- Beckers, G. and Dehez, B. (2014b). Modelling of electric field and stress in piezoelectric composite plates under bending load. In *Applications of Ferroelectrics, International Workshop on Acoustic Transduction Materials and Devices & Workshop on Piezoresponse Force Microscopy (ISAF/IWATMD/PFM), 2014 Joint IEEE International Symposium on the*, pages 1–4. IEEE.
- Beckers, G. and Dehez, B. (2017). Numerical study of an electromagnetic valveless peristaltic micropump. In *Electrimacs, 2017 IEEE International Conference*. IEEE.

- Brenner, S. C., Sung, L.-Y., Zhang, H., and Zhang, Y. (2012). A quadratic c^0 interior penalty method for the displacement obstacle problem of clamped kirchhoff plates. *SIAM Journal on Numerical Analysis*, 50(6):3329–3350.
- Chen, B., Chen, X., and Kanzow, C. (2000). A penalized fischer-burmeister ncp-function. *Mathematical Programming*, 88(1):211–216.
- Coco, A. and Russo, G. (2013). Finite-difference ghost-point multigrid methods on cartesian grids for elliptic problems in arbitrary domains. *Journal of Computational Physics*, 241:464–501.
- Dash, P. and Singh, B. (2009). Nonlinear free vibration of piezoelectric laminated composite plate. *Finite Elements in Analysis and Design*, 45(10):686–694.
- Dehez, B. (2011). Improved constitutive equations of piezoelectric monomorphs: Application to the preliminary study of an original traveling-wave peristaltic pump. *Sensors and Actuators A: Physical*, 169(1):141–150.
- Dunsch, R. and Breguet, J.-M. (2007). Unified mechanical approach to piezoelectric bender modeling. *Sensors and Actuators A: physical*, 134(2):436–446.
- Fan, C.-M., Liu, C.-S., Yeh, W., and Chan, H.-F. (2010). The scalar homotopy method for solving non-linear obstacle problem. *Computers, Materials, & Continua*, 15(1):67–86.
- Fernandes, A. and Pouget, J. (2010). Enhanced piezoelectric multilayered beam: Modelling, experimental validation and application to actuation. In *ASME 2010 Conference on Smart Materials, Adaptive Structures and Intelligent Systems*, pages 275–284. American Society of Mechanical Engineers.
- Fernandes, L. M., Figueiredo, I. N., and Júdice, J. J. (2001). On the solution of a finite element approximation of a linear obstacle plate problem.
- Fukushima, M. (1992). Equivalent differentiable optimization problems and descent methods for asymmetric variational inequality problems. *Mathematical Programming*, 53(1):99–110.
- Gopinathan, S. V., Varadan, V. V., and Varadan, V. K. (2000). A review and critique of theories for piezoelectric laminates. *Smart Materials and Structures*, 9(1):24.
- Gysen, B., Meessen, K., Paulides, J., and Lomonova, E. (2010). General formulation of the electromagnetic field distribution in machines and devices using fourier analysis. *IEEE Transactions on Magnetics*, 46(1):39–52.
- Hartmann, S., Brunssen, S., Ramm, E., and Wohlmuth, B. (2007). Unilateral non-linear dynamic contact of thin-walled structures using a primal-dual active set strategy. *International journal for numerical methods in engineering*, 70(8):883–912.

- He, X., Xu, L., Zhang, X., and Yang, S. (2016). A bidirectional valveless piezoelectric micropump with three chambers applying synthetic jet. *Journal of Mechanical Science and Technology*, 30(9):4015–4022.
- Hintermüller, M. and Laurain, A. (2010). A shape and topology optimization technique for solving a class of linear complementarity problems in function space. *Computational Optimization and Applications*, 46(3):535–569.
- Homsy, A., Linder, V., Lucklum, F., and de Rooij, N. F. (2007). Magneto-hydrodynamic pumping in nuclear magnetic resonance environments. *Sensors and Actuators B: Chemical*, 123(1):636–646.
- il Jeong, S. and Seyed-Yagoobi, J. (2002). Experimental study of electrohydrodynamic pumping through conduction phenomenon. *Journal of Electrostatics*, 56(2):123 – 133.
- Iverson, B. D. and Garimella, S. V. (2008). Recent advances in microscale pumping technologies: a review and evaluation. *Microfluidics and Nanofluidics*, 5(2):145–174.
- Jabbari, M., Ghayour, M., and Mirdamadi, H. (2016a). Dynamics analysis of the steady and transient states of a nonlinear piezoelectric beam by a finite element method. *Journal of Solid Mechanics Vol*, 8(2):247–261.
- Jabbari, M., Ghayour, M., and Mirdamadi, H. R. (2016b). Experimental and numerical results of dynamics behavior of a nonlinear piezoelectric beam. *Mechanics of Advanced Materials and Structures*, 23(8):853–864.
- Kapurja, S., Kumari, P., and Nath, J. (2010). Efficient modeling of smart piezoelectric composite laminates: a review. *Acta Mechanica*, 214(1-2):31–48.
- Kapurja, S. and Nath, J. (2013). Coupled global-local and zigzag-local laminate theories for dynamic analysis of piezoelectric laminated plates. *Journal of Sound and Vibration*, 332(2):306–325.
- Kim, J.-w., Suzuki, T., Yokota, S., and Edamura, K. (2012). Tube-type micropump by using electro-conjugated fluid (ecf). *Sensors and Actuators A: Physical*, 174:155–161.
- Kim, S.-K., Choi, J., Park, S., and Song, O. (2015). Vibration and stability of laminated composite plates subjected to a combined electromagnetic and thermal field. *Journal of Vibration and Control*, 21(12):2325–2339.
- Krommer, M. (2001). On the correction of the bernoulli-euler beam theory for smart piezoelectric beams. *Smart Materials and Structures*, 10(4):668.

- Laser, D. J. and Santiago, J. G. (2004). A review of micropumps. *Journal of Micromechanics and Microengineering*, 14(6):R35.
- Lazarus, A., Thomas, O., and Deü, J.-F. (2012). Finite element reduced order models for nonlinear vibrations of piezoelectric layered beams with applications to nems. *Finite Elements in Analysis and Design*, 49(1):35–51.
- Le, D.-V., Khoo, B. C., and Peraire, J. (2006). An immersed interface method for viscous incompressible flows involving rigid and flexible boundaries. *Journal of Computational Physics*, 220(1):109–138.
- Liu, C.-S. and Atluri, S. N. (2008). A fictitious time integration method (ftim) for solving mixed complementarity problems with applications to non-linear optimization. *CMES: Computer Modeling in Engineering & Sciences*, 34(2):155–178.
- Matar, M., Al-Halhouli, A. T., Dietzel, A., and Büttgenbach, S. (2017). Microfabricated centrifugal pump driven by an integrated synchronous micromotor. *Microsystem Technologies*, 23(7):2475–2483.
- Maurini, C., Porfiri, M., and Pouget, J. (2006a). Numerical methods for modal analysis of stepped piezoelectric beams. *Journal of Sound and Vibration*, 298(4):918–933.
- Maurini, C., Pouget, J., and Dell’Isola, F. (2006b). Extension of the euler-bernoulli model of piezoelectric laminates to include 3d effects via a mixed approach. *Computers & structures*, 84(22):1438–1458.
- Mayer, U. M., Popp, A., Gerstenberger, A., and Wall, W. A. (2010). 3d fluid-structure-contact interaction based on a combined xfem fsi and dual mortar contact approach. *Computational Mechanics*, 46(1):53–67.
- Meessen, K. J., Gysen, B. L., Paulides, J. J., and Lomonova, E. A. (2010). Three-dimensional magnetic field modeling of a cylindrical halbach array. *IEEE Transactions on magnetics*, 46(6):1733–1736.
- Mitchell, J. and Reddy, J. (1995). A refined hybrid plate theory for composite laminates with piezoelectric laminae. *International Journal of Solids and Structures*, 32(16):2345–2367.
- Nadal, C., Giraud-Audine, C., Giraud, F., Amberg, M., and Lemaire-Semail, B. (2017). Modelling of a beam excited by piezoelectric actuators in view of tactile applications. *Mathematics and Computers in Simulation*, 131:295–315.
- Nakamoto, K., Ishida, T., Kitamura, N., and Takeuchi, Y. (2011). Fabrication of microinducer by 5-axis control ultraprecision micromilling. *CIRP Annals-Manufacturing Technology*, 60(1):407–410.

- Nguyen, N.-T. (2012). Micro-magnetofluidics: interactions between magnetism and fluid flow on the microscale. *Microfluidics and nanofluidics*, 12(1-4):1–16.
- Nisar, A., Afzulpurkar, N., Mahaisavariya, B., and Tuantranont, A. (2008). Mems-based micropumps in drug delivery and biomedical applications. *Sensors and Actuators B: Chemical*, 130(2):917–942.
- Oh, K. W. and Ahn, C. H. (2006). A review of microvalves. *Journal of micromechanics and microengineering*, 16(5):R13.
- Park, J.-H., Yoshida, K., and Yokota, S. (1999). Resonantly driven piezoelectric micropump. *Mechatronics*, 9(7):687 – 702.
- Pasquariello, V., Hammerl, G., Örley, F., Hickel, S., Danowski, C., Popp, A., Wall, W. A., and Adams, N. A. (2016). A cut-cell finite volume–finite element coupling approach for fluid–structure interaction in compressible flow. *Journal of Computational Physics*, 307:670–695.
- Persson, P.-O. and Strang, G. (2004). A simple mesh generator in matlab. *SIAM review*, 46(2):329–345.
- Peskin, C. S. (2002). The immersed boundary method. *Acta numerica*, 11:479–517.
- Plagianakos, T. S. and Papadopoulos, E. G. (2015). Higher-order 2-d/3-d layerwise mechanics and finite elements for composite and sandwich composite plates with piezoelectric layers. *Aerospace Science and Technology*, 40:150–163.
- Pozzolini, C. (2009). *Continuation dans les problèmes de contact pour des plaques en flexion*. PhD thesis, Université de Provence-Aix-Marseille I.
- Reddy, J. N. (2004). *Mechanics of laminated composite plates and shells: theory and analysis*. CRC press.
- Saravanos, D. A. and Heyliger, P. R. (1999). Mechanics and computational models for laminated piezoelectric beams, plates, and shells. *Applied Mechanics Reviews*, 52:305–320.
- Scholz, R. (1984). Numerical solution of the obstacle problem by the penalty method. *Computing*, 32(4):297–306.
- Sethian, J. A. and Wiegmann, A. (2000). Structural boundary design via level set and immersed interface methods. *Journal of computational physics*, 163(2):489–528.
- Smal, O., Dehez, B., Raucourt, B., De Volder, M., Peirs, J., Reynaerts, D., Ceysens, F., Coosemans, J., and Puers, R. (2008). Modelling, characterization and testing of an ortho-planar micro-valve. *Journal of Micro-Nano Mechatronics*, 4(3):131–143.

- Smits, J. G. (1990). Piezoelectric micropump with three valves working peristaltically. *Sensors and Actuators A: Physical*, 21(1-3):203–206.
- Smits, J. G., Dalke, S. I., and Cooney, T. K. (1991). The constituent equations of piezoelectric bimorphs. *Sensors and Actuators A: Physical*, 28(1):41–61.
- Tsai, N.-C. and Sue, C.-Y. (2007). Review of mems-based drug delivery and dosing systems. *Sensors and Actuators A: Physical*, 134(2):555–564.
- Waldschik, A. and Büttgenbach, S. (2010a). Fabrication of internally driven micro centrifugal force pumps based on synchronous micro motors. *Microsystem Technologies*, 16(7):1105–1110.
- Waldschik, A. and Büttgenbach, S. (2010b). Micro gear pump with internal electromagnetic drive. *Microsystem Technologies*, 16(8):1581–1587.
- Whiteley, J. (2017). *Finite Element Methods: A Practical Guide*. Springer.
- Whitesides, G. M. (2006). The origins and the future of microfluidics. *Nature*, 442(7101):368.
- Wiegmann, A. and Bube, K. P. (2000). The explicit-jump immersed interface method: finite difference methods for pdes with piecewise smooth solutions. *SIAM Journal on Numerical Analysis*, 37(3):827–862.
- Woiias, P. (2005). Micropumps past, progress and future prospects. *Sensors and Actuators B: Chemical*, 105(1):28–38.
- Yamahata, C., Chastellain, M., Parashar, V. K., Petri, A., Hofmann, H., and Gijs, M. A. (2005). Plastic micropump with ferrofluidic actuation. *Journal of microelectromechanical systems*, 14(1):96–102.
- Yau, S.-T. and Gao, Y. (1992). Obstacle problem for von kármán equations. *Advances in Applied Mathematics*, 13(2):123–141.
- Yokota, S. (2014). A review on micropumps from the viewpoint of volumetric power density. *Mechanical Engineering Reviews*, 1(2):DSM0014–DSM0014.
- Zacks, R. (2003). Microfluidics. <http://www2.technologyreview.com/news/401770/microfluidics/>.
- Zhang, C., Xing, D., and Li, Y. (2007). Micropumps, microvalves, and micromixers within pcr microfluidic chips: advances and trends. *Biotechnology advances*, 25(5):483–514.
- Zhang, S. and Schmidt, R. (2014). Large rotation theory for static analysis of composite and piezoelectric laminated thin-walled structures. *Thin-Walled Structures*, 78:16–25.

- Zhu, D. and Wang, Q. J. (2011). Elastohydrodynamic lubrication: a gateway to interfacial mechanics review and prospect. *Journal of Tribology*, 133(4):041001.