

# ES 240 FEM Project Report

## Sloshing of a liquid in a rectangular tank

Pawel Zimoch

12/16/2008

### Abstract

This report summarizes my work on Finite Element Method analysis project for Engineering Sciences 240.

The initial goal of the project was to analyse sloshing of a liquid in a cylindrical tank. Sloshing in general has great practical significance, especially in transportation and storage tanks. I gained personal interest in sloshing in cylindrical tanks in particular after performing a simple experiment with a cup of coffee. I was especially interested in finding out the resonance frequency of the forced oscillation of the liquid under horizontal harmonic forcing.

I decided to use the COMSOL Multiphysics software package due to its support of moving meshes which allows for modeling of free fluid surfaces. After several initial failed attempts at using 3D geometry, I settled for a 2D, rectangular subdomain. I used the COMSOL's demonstration model as a basis for my analysis.

I analyzed the response of glycerine in a 1m by 0.3m rectangular tank subject to constant deceleration on  $1 \text{ m/s}^2$  for two seconds, after which the only force acting on the fluid was its own weight. The parameters analysed were the total horizontal force on the vertical boundaries of the tank, vertical displacement of fluid on vertical boundaries and pressures at three points on the vertical boundaries, as a function of time.

I found that the dynamical inertial load of a fluid under deceleration exceeds that of a solid of the same density by about 50%. Maximum vertical displacement on both boundaries was approximately 0.11m. Pressure distribution on the vertical boundaries also differs from a stationary case, pressure in the upper of the tank being slightly higher.

The Reynolds number for the analysed system is approx. 180, and thus the fluid can be modeled as approximately inviscid. This approximation, however, has some limitations, which this analysis was unable to overcome. Moreover, this analysis has very limited application to any real physical system and its outcomes should be used with great caution.

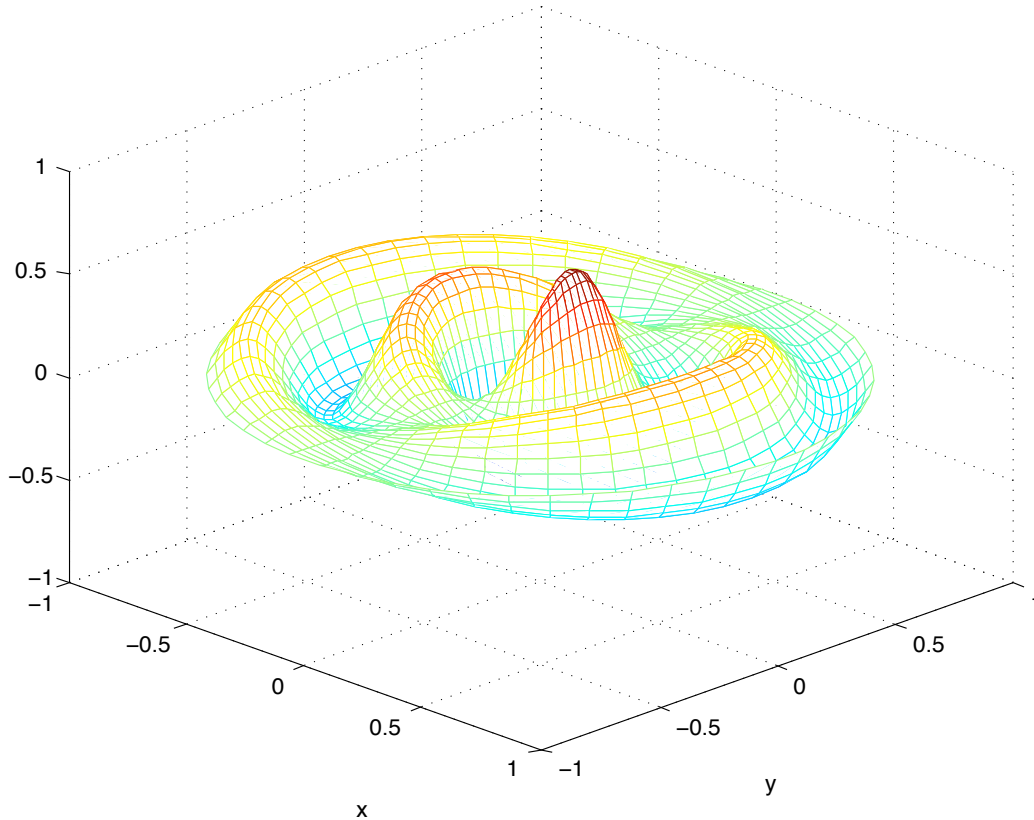


Figure 1: One of a vibrating drum's mode shapes

## 1 Introduction

Analysis of the sloshing motion of a contained liquid is of great practical importance. Motion of a fluid can persist beyond application of a direct load to the container, the inertial load exerted by the fluid is time-dependent and can be greater than the load exerted by a solid of the same mass. This makes analysis of sloshing especially important for transportation and storage tanks. Due to its dynamic nature, sloshing can strongly affect performance and behaviour of transportation vehicles, especially tankers filled with oil. In fact, a significant amount of research has gone into developing numerical models for predicting fluid behaviour under various loads (put a reference in here).

My interest in the topic began when I realized that oscillations of coffee in a mug I was holding were dependent on my walking pace. This sparked my curiosity and I performed a simple experiment in which I oscillated the mug horizontally at different frequencies. I realized that different frequencies produced different mode shapes on the surface of the coffee. In particular, I noticed shapes similar to mode shapes of a vibrating drum, obtained by the use of Bessel functions (Fig. ??<sup>1</sup>).

I started thinking how such forced oscillations depended on forcing frequency. Since the system was too complicated for me to analyse, I could not reach a simple conclusion. Therefore, I decided to numerical methods to analyze the problem, and this project seemed a great opportunity to use Finite Elements Method to help me understand the phenomenon.

---

<sup>1</sup>Frame from an animation prepared by Shreyas Madre for the purpose of the course Applied Mathematics 105b at Harvard College in Spring 2008



Figure 2: Basic model's subdomain

## 2 Basic modelling approach

### 2.1 Software choice

Having virtually zero experience with FEM software I was unaware of the limitations of its different kinds, and I didn't pay attention to them when I was choosing the topic. This proved to have been a mistake, since I soon realized that performing the analysis I wanted to do would be impossible using ABAQUS, since modelling fluid-solid interactions wasn't one of its strengths. I was told, however, that COMSOL Multiphysics could be better for my purpose, since it supports a "moving mesh" feature, where the mesh deforms with the fluid, hence allowing accurate representation of free surfaces. I was also shown that the standard version of COMSOL featured a demonstration model of liquid sloshing in a 2D rectangular tank. Given this information, I decided to focus on COMSOL and not investigate further into capabilities of ABAQUS.

### 2.2 COSMOL's model

#### 2.2.1 Geometry

The model provided by COMSOL is a 2D model of a rectangular tank filled with glycerol ( $\rho = 1270 \text{ kg/m}^3$ ,  $\mu = 1.49 \text{ Pa s}$ ), sloshing due to oscillations of the gravity vector, equivalent to tilting the tank with amplitude of  $4^\circ$  and frequency of  $1 \text{ Hz}$ .

The model is based on a body of fluid comprising of a rectangular domain 1m long and 0.3m high, representing a fluid obeying Navier-Stokes equations for transient, incompressible flow. This is represented in Fig. (put reference here), where x-direction is along the horizontal axis and y-direction along the vertical axis.

On top of that subdomain was superimposed an automatically generated, moving mesh, represented in Fig. ??.

#### 2.2.2 Boundary conditions

Since there are two discrete elements in the model (Navier-Stokes subdomain and moving mesh), there must be two sets of boundary conditions.

For the Navier-Stokes subdomain, the vertical boundaries and the bottom of the tank are defined as

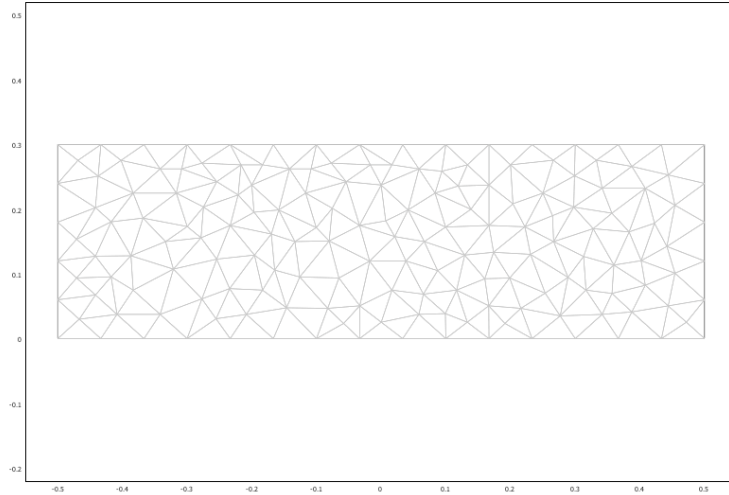


Figure 3: Basic model's mesh

slip/symmetry boundaries. That is, at the surface

$$\vec{u} \cdot \vec{n} = 0 \quad (1)$$

holds, where  $\vec{u}$  is the fluid velocity and  $\vec{n}$  is the vector normal to the surface. Top boundary is a neutral boundary, at which the stress is zero.

For the moving mesh, boundary conditions must meet the deformation of the subdomain. Since the subdomain is supposed to simulate a tank, the vertical boundaries' displacement in the x-direction is set to 0, but the mesh is allowed to move in the y-direction. The movement on the bottom boundary is restricted in both directions. The boundary condition on the top boundary is more complicated: deformation of the mesh must meet the deformation of the free surface. Therefore the velocity of the mesh in the direction normal to the mesh must be equal to the velocity of the fluid in the direction normal to its surface. The initial condition is set up so that the mesh fits the fluid perfectly at the beginning of the motion. This could be expressed as

$$\vec{\psi} \cdot \vec{n}_m = \vec{u} \cdot \vec{n}_f \quad (2)$$

where  $\vec{\psi}$  and  $\vec{u}$  are the velocities of the mesh and the fluid on the top surface, and  $\vec{n}_m$  and  $\vec{n}_f$  are vectors normal to the mesh and fluid surface, respectively.

## 2.3 First models

### 2.3.1 3D Cylindrical subdomain

Since I initially intended to simulate sloshing in a cylindrical tank, I decided to use approach similar to the one shown above to a 3D cylindrical subdomain. I created a solid cylinder of radius 0.1m and height 0.1 (fig. ??) and applied identical boundary conditions as above to the subdomain and the moving mesh (extended to 3D). Similarly to the built-in model, I enforced oscillation with a rotating gravity vector  $\vec{g}$ :

$$g_x = 0.01 \sin(2\pi t) \times \rho \quad (3)$$

$$g_y = 0 \quad (4)$$

$$g_z = 9.81 \times \rho \quad (5)$$

where 0.01 is the amplitude of vibration,  $2\pi$  the radial frequency,  $\rho$  the density of the fluid and z-direction is along the vertical axis. This however, did not yield expected results as model was underconstrained. Despite my best efforts at trying to fix the problem, I was unable to obtain a solution.

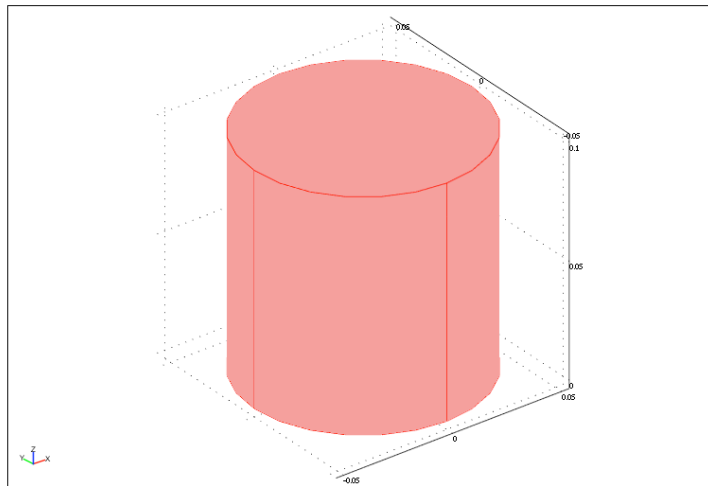


Figure 4: 3D cylindrical subdomain

### 2.3.2 3D Rectangular subdomain

Seeing that cylindrical subdomain was beyond my understanding, I decided to “step down” to a less complicated rectangular domain (fig.??). Again, I set up boundary conditions exactly as in the COMSOL’s demonstration model. I used a forcing condition identical as in the cylindrical subdomain case. This time the problem was no longer underdetermined, but the solver could not resolve the boundary constraints, and I couldn’t obtain a solution.

### 2.3.3 2D Rectangular subdomain

Afer the failures with 3D cylindrical and rectangular subdomains, I decided to use a 2D rectangular subdomain, presented in Fig. (put in a reference). To simulate horizontal oscillations, I imposed the following forcing function:

$$g_x = 0.01 \sin(2\pi t) \times \rho \quad (6)$$

$$g_z = 9.81 \times \rho \quad (7)$$

However, I was not able to obtain clear picture of resonance of the liquid in the range of frequencies 1 – 10 Hz because the displacement of the fluid was so large that it failed to converge.

I also realized that since this situation was so far away from my initial goal and I still failed to get any results, I should settle for a simpler problem to which I can attempt to find a solution.

## 3 Final Model

After the intial failed attempts at obtaining useful results, I decided to model a rectangular tank in 2 dimensions, experiencing constant deceleration of  $1 \text{ m/s}^2$  in the x-direction (or equal acceleration in the -x-direction). The model used was the same as the one described in section 2.3.3. The model parameters are summarized inthe table below:

parameter	value
height of subdomain	0.3 m
length of subdomain	1.0 m
density	1270 kg/m <sup>3</sup>
dynamic viscosity	1.49 Pa s

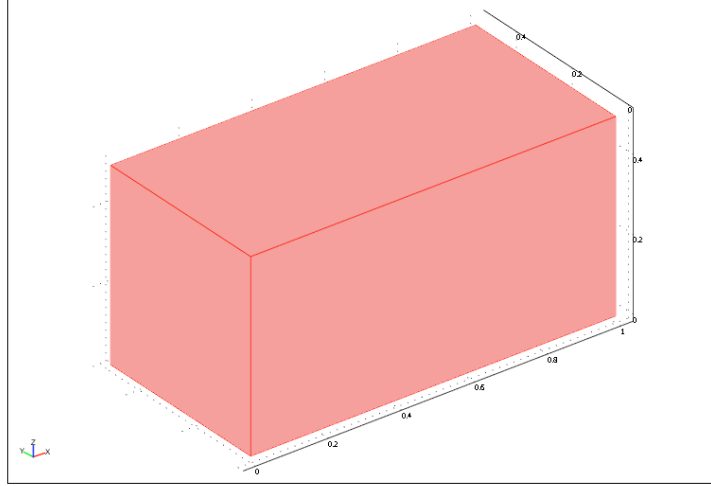


Figure 5: 3D rectangular subdomain

Boundary Conditions (Navier-Stokes):

boundary	expression
left	slip/symmetry $\vec{u} \cdot \vec{n} = 0$
bottom	slip/symmetry
right	slip/symmetry
top	neutral (no force)

Boundary Conditions (Moving Mesh):

boundary	expression
left	$\vec{d}_x = 0$
right	$\vec{d}_x = 0$
bottom	$\vec{d}_x = \vec{d}_y = 0$
top	$\vec{\psi} \cdot \vec{n}_m = \vec{u} \cdot \vec{n}_f$

where  $\vec{d}_x$  and  $\vec{d}_y$  are x and y displacements of the mesh,  $\vec{\psi}$  and  $\vec{u}$  velocity vectors of the mesh and fluid on the top boundary and  $\vec{n}_m$  and  $\vec{n}_f$  the vectors normal to the mesh and fluid on the top boundary, respectively.

The imposed forcing condition was :

$$g_x = 1 \times \rho \times u(t - 2) \quad (8)$$

$$g_z = 9.81 \times \rho \quad (9)$$

where  $u(t)$  is the Heaviside step function.

Given this model, my goal was to find the following physical quantities:

- total force per length exerted by the fluid on the vertical boundaries of the tank in the x-direction as a function of time
- total height of fluid on the vertical boundaries as a function of time

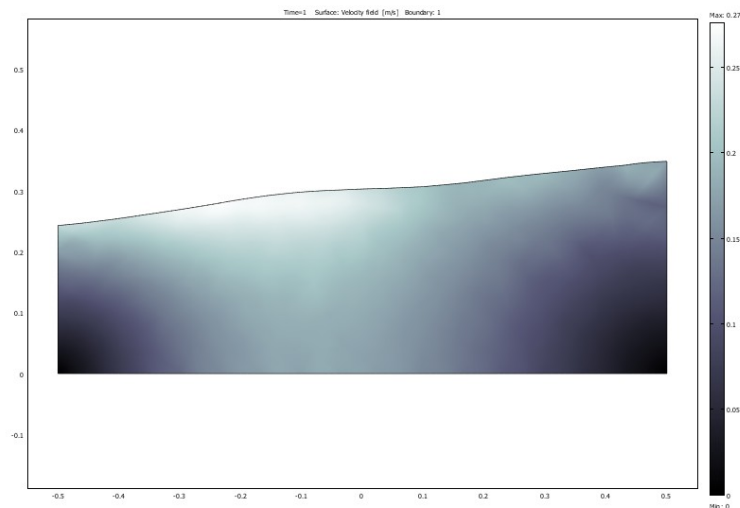


Figure 6: Solution 1s after deceleration began

- pressure at  $x=0.0$  m,  $x=0.15$  m and  $x=0.3$  m on both vertical boundaries as a function of time

The above parameters are only some of the data generated by the simulation. I picked them because I believe they would constitute the most important information necessary to an engineer designing the tank or the transporting vehicle. However, I have never attempted such design, not have I researched the field sufficiently to state that with an appreciable degree of confidence.

## 4 Results

The simulation provided a visual output in form of an animation, which was shown during the presentation. Since this is impossible to reproduce here, below are several static frames from the animation, taken at times which give a visual idea of the sloshing. The color scale on the right-hand side of the graphs represents magnitude of vorticity.

Figure ?? shows the solution 1s after the deceleration began.

Figure ?? shows the solution right before deceleration stops.

Figures ?? and ?? show solutions at 4.7s and 7.2s after the deceleration has ended.

However, despite its visual appeal, the animation does not provide useful data on the system. More useful information is provided by the following graphs.

Fig. ?? shows the total force on the horizontal walls in the x-direction as a function of time.

Fig. ?? shows the displacement of the fluid on the right boundary as a function of time.

Fig. ?? shows the displacement of the fluid on the left boundary as a function of time.

Fig ?? shows pressure at various points on the left boundary as a function of time.

Fig ?? shows pressure at various points on the right boundary as a function of time.

## 5 Discussion

Despite not having performed a theoretical analysis of the system before performing the simulation, I will attempt to perform a basic analysis of the data. I will also try to present it in a more meaningful form, since the graphs in section 3 do not give the information in an easy-to-digest format.

Perhaps the most important information provided by the model – force in the x-direction on both horizontal boundaries – is contained in Fig. ?. It is clear that during the deceleration period the

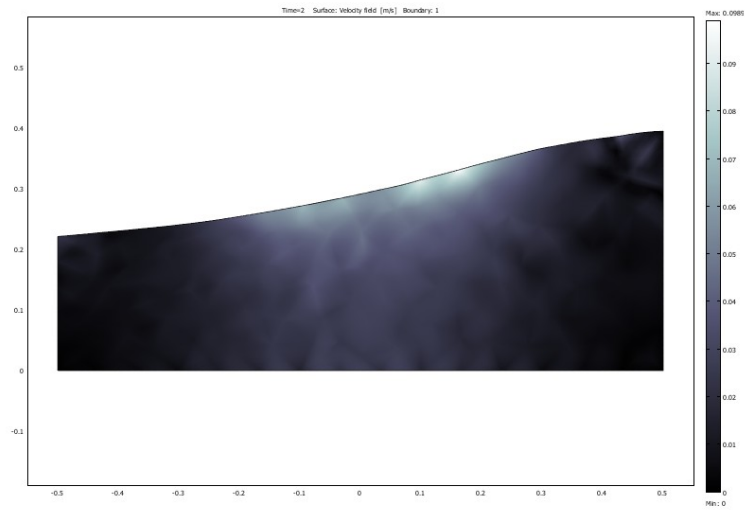


Figure 7: Solution right before deceleration stops

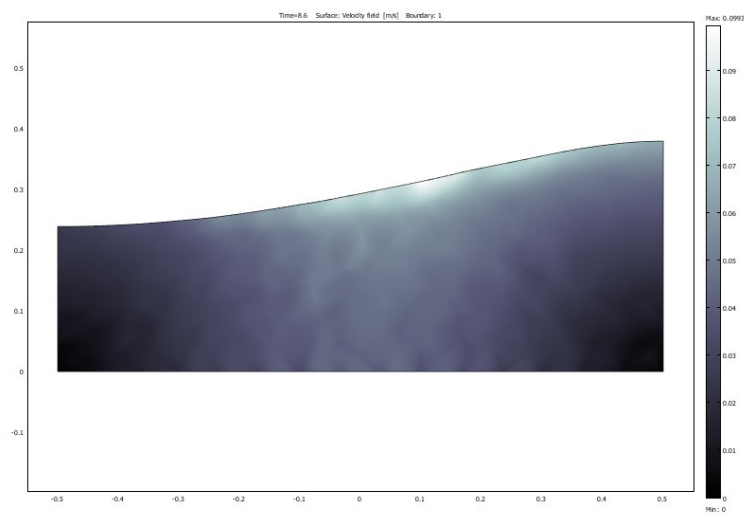


Figure 8: Solution at 6.7 s

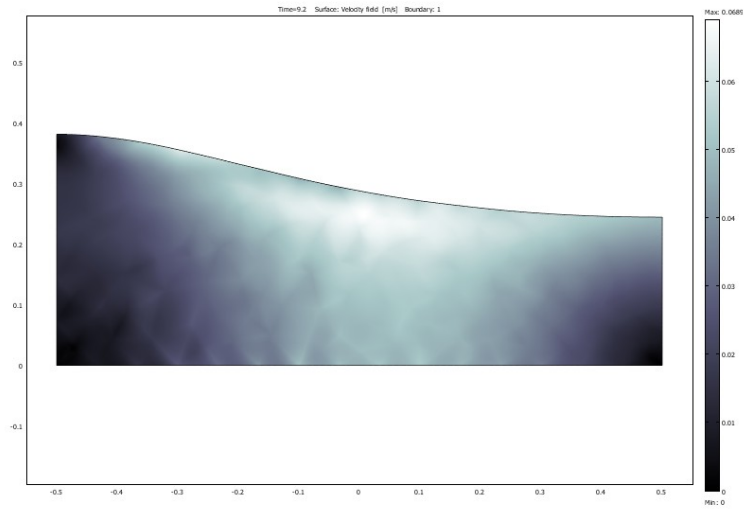


Figure 9: Solution at 9.2 s

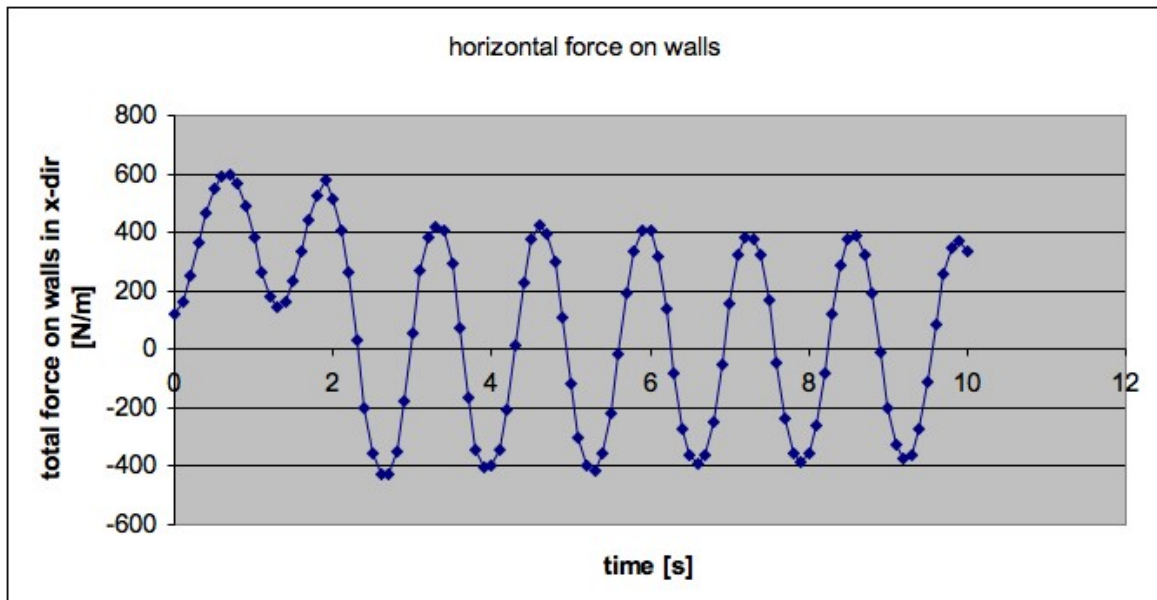


Figure 10: Force on horizontal walls in the x-direction

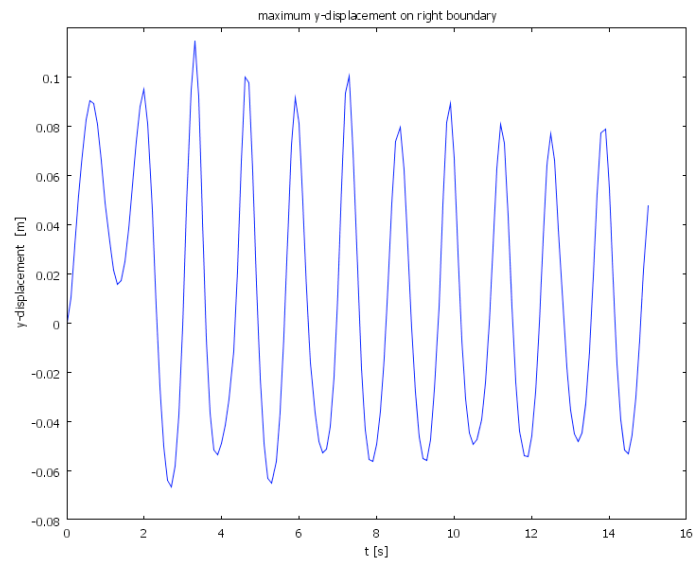


Figure 11: Displacement of fluid on the right boundary

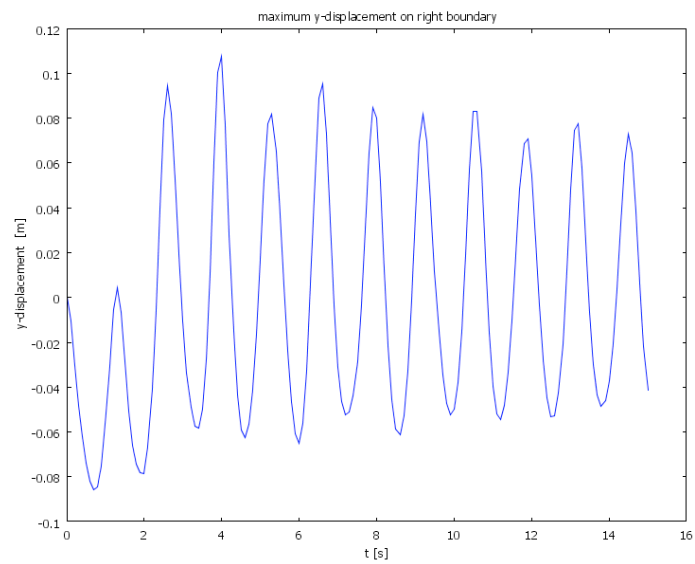


Figure 12: Displacement of fluid on the left boundary

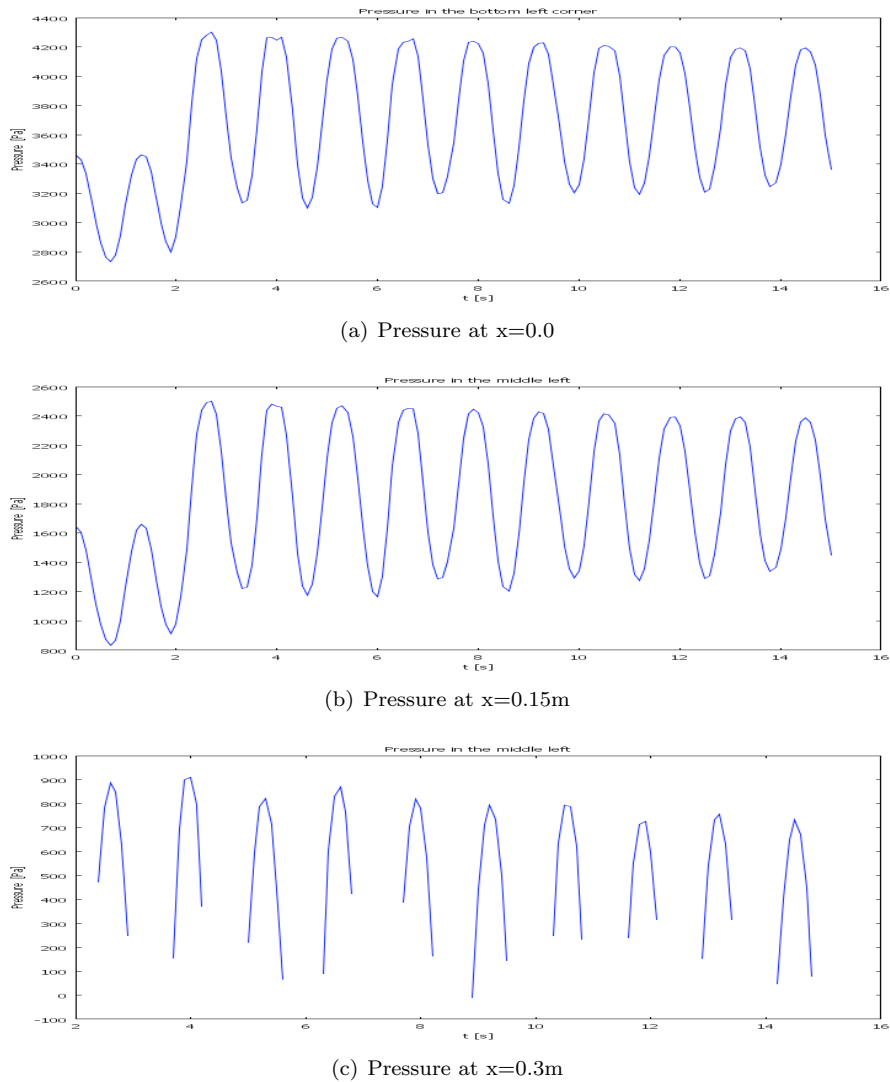


Figure 13: Pressures on the left boundary

force reaches 600 N per unit width of the tank for two brief periods. If the fluid were replaced by a solid with identical density ( $\rho = 1270kg/m^3$ ), its inertial load per unit width of the tank would be

$$F_x = 1.0 \times 0.3 \times 1270 \times 1 = 381N/m \tag{10}$$

Therefore, The inertial load of the analysed fluid exceeds that of a solid by over 50%. After the deceleration ceases, the fluid exerts a force which oscillates with an amplitude of 400 N/m, which is as much as a solid of the same density would provide.

Another piece of useful information is the vertical displacement of the fluid on the vertical boundaries, presented in figures ?? (right boundary) and ?? (left boundary). This could be especially important in the case of open tanks, where spillage is undesirable, or when for whatever reason too large vertical displacement should be prevented. On both boundaries maximum displacement reaches approx. 0.11m above the initial position. Since the initial depth of fluid was 0.3m, maximum vertical displacement constitutes  $\approx 35\%$  of the initial depth. However, a conclusion that this result can be generalized to tanks of different physical dimensions would be unjustified. This could be established by performing a set of additional simulations, or possibly by a scaling argument, not attempted here.

The last important (I think) piece of data is the distribution of pressures on the vertical boundaries as a function of time. Pressures at three representative points: bottom ( $x=0m$ ), middle ( $x=0.15m$ ) and top ( $x=0.3m$ ) of the initial subdomain on the left and right boundaries are plotted against time in figures ?? and ??, respectively. It should be noted that both during the period of deceleration

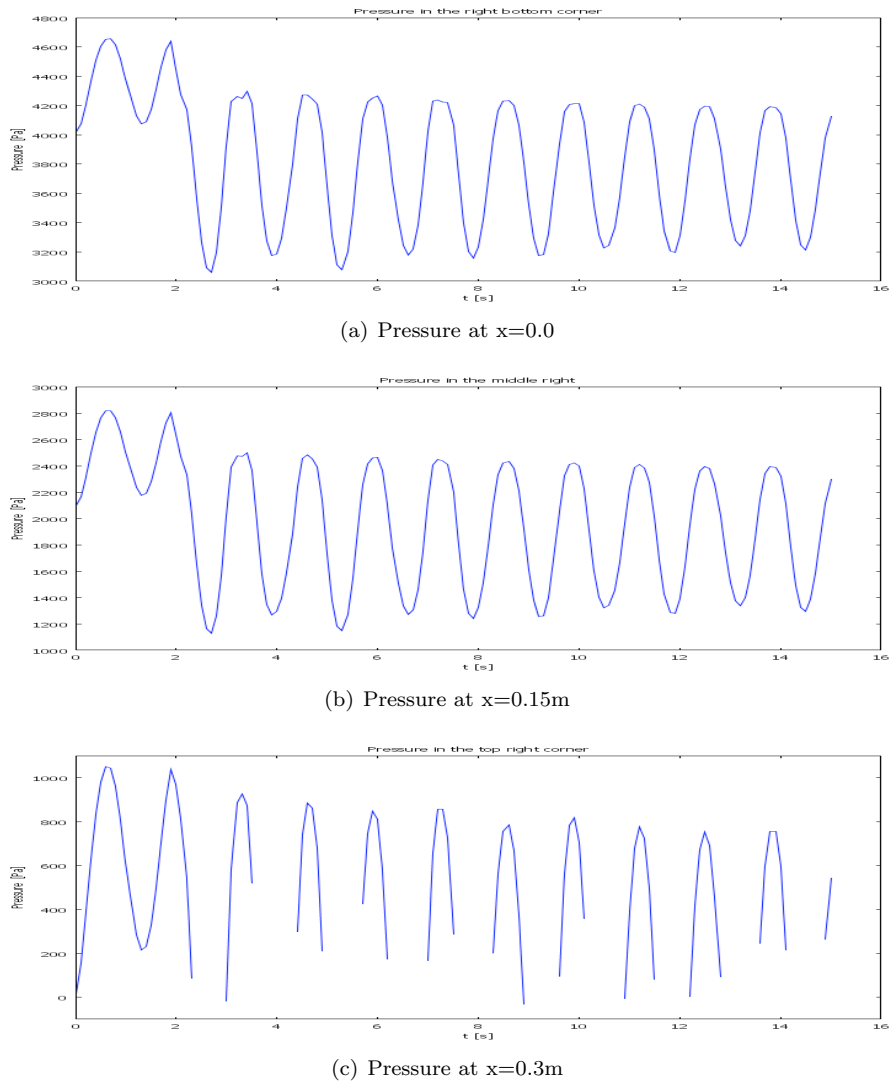


Figure 14: Pressures on the left boundary

and the period of oscillation, pressure at  $x=0.15\text{m}$  was approx. 60% of the pressure at  $x=0.0\text{m}$ . When contrasted with the percentage for static fluid (50%) it can be concluded that there is a fairly significant increase in pressure, resulting from the horizontal movement of the fluid. The graphs in figures ?? and ?? are discontinuous because the fluid was not always in contact with the boundary at this point during sloshing.

## 6 Evaluation

Whenever a solution is obtained by the means of numerical computation, applicability and validity of the model must be assessed. This section aims at addressing some basic issues related to the validity of this model and the extent to which it represents the actual physical system.

The characteristic of the model which attracted my attention immediately after the simulation, as well as the attention of the audience after the presentation was the apparent lack of damping present in the liquid. As evidenced by figures ?? – ??, amplitudes of force, vertical displacement as well as pressure decrease only slightly in the first 13 seconds after the deceleration ceased. This is contrary to the basic intuition for behaviour of glycerol (fluid modeled). For viscous flows, viscous friction at the boundary can be one of the main sources of energy dissipation (although energy is dissipated due to viscous friction within the fluid as well), and so the results put into question the validity of the slip

boundary condition on the Navier-Stokes subdomain. To model the behavior of a fluid away from the boundary, slip boundary can be used for flows characterized by high Reynolds numbers, defined as

$$Re = \frac{\rho U L}{\mu} \quad (11)$$

where  $\rho$  and  $\mu$  are the density and dynamic viscosity of the fluid,  $U$  is the characteristic speed of the flow and  $L$  its characteristic length. For the flow modeled here,  $\rho = 1270 \text{ kg/m}^3$  and  $\mu = 1.49 \text{ Pa s}$ . There are multiple speed and length scales that could be defined. I think, however, that when evaluating the slip boundary condition, it is most reasonable to define the characteristic speed as the average vertical speed at the boundary, and the characteristic length as the length of the tank (1m). Average vertical speed can be calculated by dividing the peak-to-peak amplitude of vertical displacement on one of the boundaries (0.16m from fig. ??) by half of the oscillations' period ( $\approx .075 \text{ s}$  from figs. ?? – ??), which yields characteristic speed of  $0.213 \text{ m/s}$ . Defined on such criteria, Reynolds number is

$$Re = \frac{1270 \times 0.213 \times 1}{1.49} = 180 \quad (12)$$

Such value of  $Re$  signifies that the inertial effects in the modeled flow are approx. two orders of magnitude greater than viscous effects. Therefore, despite being contrary to the basic intuition, due to high density of glycerol and large scale of geometrical features, the flow modeled can be approximated as inviscid, and the no-slip condition is justified. On the other hand, this value of Reynolds number is not very high, and it should be noted that the inviscid flow approximation used in this case will have adverse effects on the correlation of results achieved here with actual data (if such has ever been/will ever be collected). Therefore, the results presented here should be used very cautiously until other analyses/experiments confirm their validity.

Of course, it should be born in mind that the entire point of numerical analysis is to achieve results as close to exact solution as possible. Therefore, having to use an approximation like this in a numerical simulation, especially for flows with Reynolds numbers in the range where use of inviscid flow approximation is debatable (as is the case here), is unfortunate, as it obstructs the very point of such analysis. To the best of my knowledge (and I must stress here that this knowledge is not great) current FEM software still struggles with resolving the no-slip boundary conditions for flows with moderate and high Reynolds numbers, due to its inability to deal with boundary layers.

On a more general note, it should be pointed out that the analysis carried out here is only applicable to the specific system: glycerol in a rectangular tank of specified dimensions sloshing under the prescribed forcing. As such, the analysis does not offer any insight into the sloshing phenomenon in general. This is due to the inherently flawed methodology of using FEM for scientific research in this case. When given the project assignment I thought that this would be a good opportunity to look at a system I did not understand, and try to use FEM to gain insight into its behavior. For this reason I chose to investigate the sloshing of a liquid in a cylindrical tank. While working on this project, I quickly realized that given the complexity of the system it was impossible so much as model it without having a concrete understanding of its behaviour — I think that my failure to obtain solutions for the initial 3D models mentioned in sections 2.3.2 and 2.3.3 was due to my inability to describe the model, rather than the software's inability to deal with simple 3D geometry. When the simple and rather naive approach of trying to “extrapolate” 2D boundary conditions to 3D failed I simply didn't know where the problem was, so I was forced to retreat back to 2D.

Rather than using the FEM right away, I should have done much more background research on the problem, and see if I can get any basic insight using previously obtained results, and simple analysis, e.g. scaling arguments, etc. Then, I should have used FEM to see if numerical solution agrees with my predictions. The analysis should have been carried out with emphasis on obtaining results which would represent the system in a much more general way, eg. across various geometric scales, or across a range of initial conditions.

Additionally, the scope of my initial goals extended vastly beyond both my knowledge and time resources. Therefore I had to narrow it down somewhat ad-hoc while working on the project, which adversely affected its focus and applicability. This can be signified by how far the results of my final analysis are from what I initially intended on doing.

In short, my approach was “let me put this in a computer and see if I can get anything useful out of

it”, rather than “let me see if the numerical computation corroborates my theoretical analysis, and if I can get any additional insight from any discrepancies that arise”.

Overcoming this naive impression of FEM and its value and applicability in gaining insight into physical phenomena is the most important lesson I learned while working on the project. When using FEM in the future, I will surely pay more attention to theoretical analysis of the system beforehand, which will allow me to make much better use of the power of numerical simulation.

Finally, I would like to point out that trying to analyse a complicated system while learning the software does not contribute to the overall productivity. Therefore, I would like to thank Ben Jordan for helping me out with using COMSOL, which made the task of figuring out the software’s inner works considerably less burdensome.

## References

- [1] Carlou, Alain and Guido Casella. 1999. Liquid Sloshing in ship tanks: a comparative study of numerical simulation. *Marine Structures*. Vol. 12, issue 3 (April), 183-198.
- [2] Liu, Dongming and Pengzhi Lin. 2008. A numerical study of three-dimensional liquid sloshing in tanks. *Journal of Computational Physics*. Vol. 227, 8 (April), 3921-3939.
- [3] Kidambi, R. and P.N Shankar. 2004. The effects of the contact angle on sloshing in containers. *Proceedings of the Royal Society*. 460,
- [4] Spyrou, K.J. and J. M. T. Thompson. 2000. The Nonlinear Dynamics of Ship Motions: A Field Overview and Some Recent Developments. *Philosophical Transactions: Mathematica, Physical and Engineering Sciences*. Vol. 358, No. 1771, The Nonlinear Dynamics of Ships, 1735-1760