POLITECNICO DI TORINO

Department of Environment, Land and Infrastructure Engineering

Master of Science in Petroleum and Mining Engineering

VIBRATION DUE TO THE DEMOLITION OF A BRIDGE STRUCTURE.

Supervisor:

Prof. Alberto Godio

Co-Supervisor:

Prof. Claudio Oggeri

Prof. Pierpaolo Oreste

Daniel Cualbu

APRIL 2019

Sommario

1) Introduction

The thesis focuses on the analysis and modelling of vibrations propagation due to the impact on the ground of a mass of demolition (by explosives) of a highway viaduct pillar; we focus also on the vibration's effects on the underground structures (which are principally pipelines). To understand the dynamics of the problem is necessary to take into account the structure's mass and volume distributions, its kinematics during falling down and the analysis the different possible modalities of impact of the different parts which play a role.

We analyze the phenomena considering the following main step: kinematics of the masses, impact on the ground, partition of energy at the impact and propagation of seismic phenomena.

After the impact, all the kinetic energy of the mass impacted on the ground is converted in other forms of energy. Part of this energy is dissipated as heat and sound; the other part is converted in vibration and so, in elastic energy. Another part is involved in elastoplastic ground transformation.

The amount of energy which is converted into elastic energy is propagated though the ground under the pillar and depends on the geophysical and geomechanical characteristics of the ground. This is the amount of energy of our interest. Hence, it is necessary a previous and accurate characterization of the soils and the structures below the pillar. Once done the characterization is possible to study and analyze the propagation of the vibrations.

The analysis of the phenomenon is developed in two principal ways. The former is to approach the real problem with an existing physical model. For what concerns the propagation of vibrations the impact with the ground is simulated through the spring-dumper model, in particular with Kevin-Voigt and Maxwell models. We solve the resulting differential equation is analytically form in order to obtain the magnitude of the displacement and the vibrations involved in the ground during and after the impact. For what concerns the effects of the vibrations on the underground structures a roughly approach is proposed. It implies the Kirsh solution for an anisotropic distribution of the underground stresses on a cylindrical hole. The latter is the study of the vibratory problem through the numerical analysis. Thus, the approximation of our real problem with a 3D numerical model. the numerical simulation is performed firstly in completely elastic media. this implies the possibility to model it with the tool "Structural Mechanics" of Matlab. We simulate the geometry of the problem, model the phenomenon with the necessary boundary conditions and initial conditions, create a refined mesh and then interpret the obtained results. Then, the results are checked with those obtained with the analytical model. The influence of the vibrations on the underground structures is firstly developed with the same approach of before. To take into consideration the viscoelastoplasticity of the problem we move to a more refined model. This time the simulation is done through the help of Flack. All the studied method is applied on a real case concerning the demolition of a reinforced concrete pillar demolition.

2) Physical characterization of the problem

In this chapter we will caracterize the problem. It is subdivided in three main parts, which are:

1. Pillar characterization
2. Undergroung pipeline characterization
3. Ground characterization

In the first paragraph we are going to define the geometry of the bridge, its parts and its nomenclature. Then, we focus on a single pillar of the bridige. Again, we explain the parts composing the pillar, and we calculate the volume which play a role in the physics of the problem. The calculation of the volume is done to estimate the mass of the body. We are going to perform it in two ways, one analytical and the other with a CAD.

The second part consist in the description of the underground works. Under the bridge, at 2 meters depth there are services pipelines, we need to characterize them and the boxes in which they are contained, in order to forcast their possible demage.

The last part is about the ground properties. An overview on the geophysical properties of the different layers, density and ground and the velocity of propagation of the waves. Then the extimation of the elastic properties of each layer, Young modulus, poisson coefficient and Shear modulus. Furthermore, the explaination of the physical behaviour of the groun when an heavy mass impact on it. The extimation of the impact pressure and the impact energy. Thanks to the heavy tamping theory it will be hypotized the time of impact.

2.1) Pillar characterisation

In figure 1 we see a general 3D prospect of a bridge. In the image we can see there are three for stay cable pillars and one without cables. Firstly, we develop a model for the pillar without cables. The model consists in the preliminary calculation of its own volumes and weights. We are going to do it with two different approaches, both of them must converge into the same results or, at least, a comparable result. This calculation will be later compared with the data furnished from real measurements. Isolating the pillar without cables we obtain that in the figure 2. We may identify two different parts by which the pillar is formed. The upper part, called deck and the lower part supporting the deck, called pole.

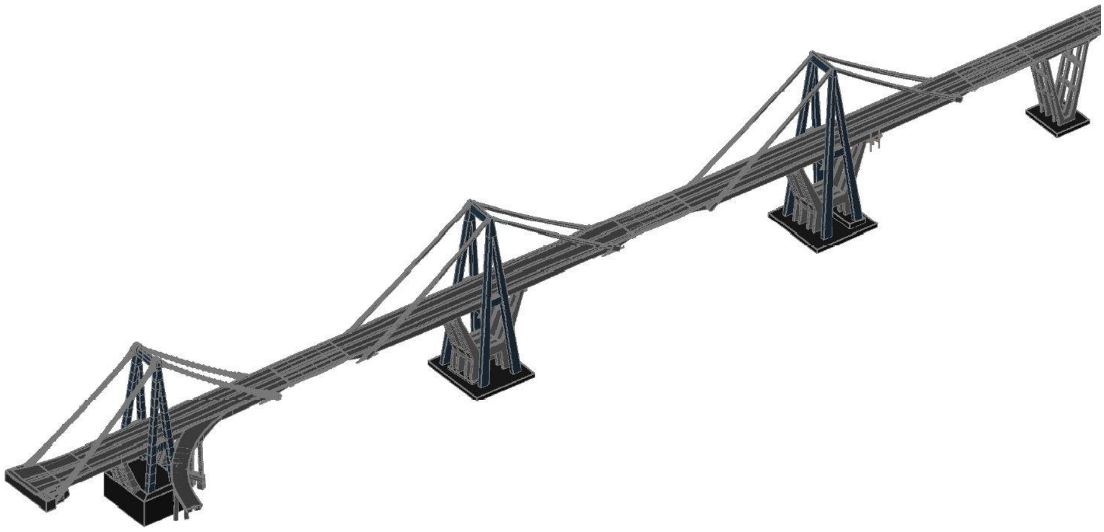


Figure 1 In the picture is represented a common viaduct with a combination of for stay cable pillars and normal pillars design.

To calculate the volume of the deck, which is the heaviest part of the pillar, we may follow two ways. The first solution is by dimensioning it. The result of the dimensioning is shown in figure 3. From the picture we can appreciate the width, the length and di height of the deck. Respectively they are 15 meters, 42 meters and 2,65 meters. The volume calculation will be conducted as follows.

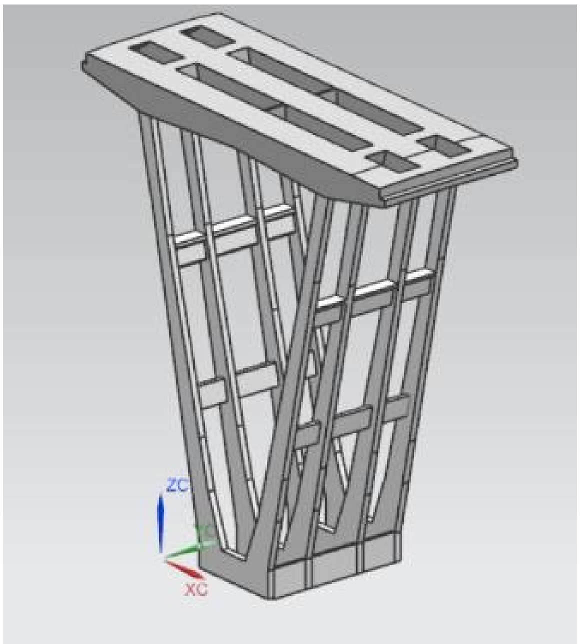


Figure 2 3D model of a pillar without cables, the upper part is called deck, the sustaining part is the pole. The 3D elaboration is done with SIEMENS

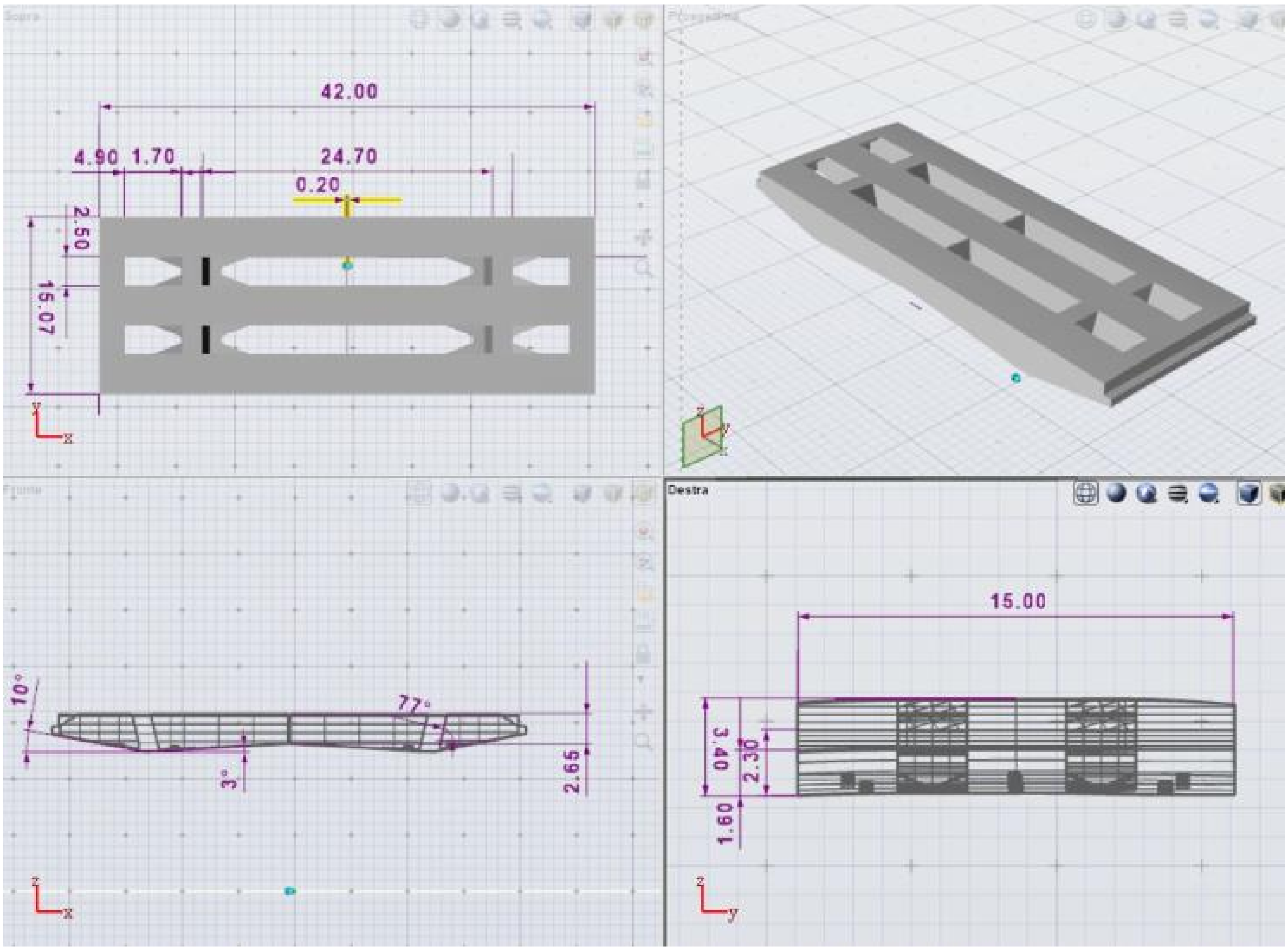


Figure 3 The orthogonal projection of the deck. The quotation is expressed in meters. The upper right part represents the deck itself. The upper left represents the above view (y-x cross section). The lower left the z-x cross section. The lower right the z-y cross section.

First the volume calculation of the parallelepiped approximating the deck, taking into account it as a solid body. Second subtracting from it the volume of the empties. From the figure 3 we may appreciate the empties, they are 4, two bigger and other fours little. The biggers have 24,7 m of length and 2,5 m width. The littles are 4.9 meters long and 2,5 meters width. The heights are the same for all and even for the rest of the structure. We will obtain:

$$V_{empties} = [2 * (w * H * L)]_1 * [4 * (w * H * L)]_2 =$$
$$= (2 * 2.5m * 2.65m * 24.7m) + (4 * 2.5m * 2.65m * 4.90m) = 168m^3$$

$$V_{parallelepiped} = w * H * L = 15.07m * 2.65m * 42m = 1677m^3$$

$$V_{tot} = V_{parallelepiped} - V_{empties} = 1677m^3 - 168m^3 = 1509m^3$$

Our deck is supposed to be made by reinforced concrete, to have an idea about its density we look in the literature. In table 1 are summarized the principal values of the concrete densities coming from the literature.

Table1: In the following table are reported some of the most common concreate densities in accordance with its chemical composition and with the bibliography.

Bibliographic Entry	Result (w/surrounding text)	Standardized Result
Concrete Basics. Portland Cement Association.	"Density in Place: Density of normal CLSM in place typically ranges from 90 to 125 pounds per cubic foot (1840 to 2320 kg/cubic m)."	2320 kg/m^3 (conventional)
Conversion Factors, Material Properties and Constants, Edward Boyden, MIT.	"Density Concrete 2242 kg/m^3"	2242 kg/m^3 (conventional)
Material Notes. Faculty of the Built Environment, University of New South Wales.	"While conventional concrete has a density of about 2300 kg/m^3, lightweight concrete has a density between 160 and 1920 kg/m^3."	2300 kg/m^3 (conventional) 160–1920 kg/m^3 (lightweight)
Cube Competition. Concrete Society of Southern Africa.	"The concrete must be of a lightweight nature, with a mass of not greater than 1.5 kg/100 mm cube or a density of 1500 kg/m^3"	< 1500 kg/m^3 (lightweight)
Baldwin, Kelly. Electrically Conductive Concrete: Properties and Potential. *Construction Canada*. Vol. 98, No. 1 (January/February 1998): 28-29.	"Density (kg/m3) 1450–1850"	1450–1850 kg/m^3 (conducting)
Concrete Admixture 453. Pt. Union Ajidharma.	Density (kg/m^3) Without 453: 2320 With 453: 2320	2320 kg/m^3 (conventional)
Transit Mix Perlite Concrete. Schundler Company.	Wet Density Range in kg/m^3 808.0 +/- 48.0 728.0 +/- 48.0 648.0 +/- 48.0 584.0 +/- 48.0	648–808 kg/m^3 (lightweight)
Aerated Concrete, Lightweight Concrete, Cellular Concrete and Foamed Concrete. Pan Pacific Management Resources.	Density 300-600 kg/m3 (19-38 lbs/ft3) Made with Cement & Foam Only Density 600-900 kg/m3 (38-56 lbs/ft3) Made with Sand, Cement & Foam Density 900-1200 kg/m3 (56-75 lbs/ft3) Made with Sand, Cement & Foam Density 1200-1600 kg/m3 (75-100 lbs/ft3) Made with Sand, Cement & Foam	300–1600 kg/m^3 (lightweight)
Is anyone buried in Hoover Dam? Bureau of Reclamation, US Department of the Interior.	"Typically, concrete has a density of 150 pounds per cubic foot, which means that a block of concrete that is one foot wide, one foot long, and one foot high would weigh 150 pounds. Water has the density of only 62.4 pounds per cubic foot."	2400 kg/m^3
Pavement Conversion Factors. Washington State Department of Transportation.	pcf / kg/m 2 PCCP: 150 / 2403 ACP: 137/0.10' / 2439	2403–2439 kg/m^3

We can look the average density is around 2400 kg/m^3, for safety reasons and to be more conservative in the calculation we take a higher value, so ρ=2500kg/m^3. Once fixed the density and found the volume of the deck it is possible to calculate the total mass:

$$m = V_{tot} * \rho = 1509m^3 * 2500\ kg/m^3 \cong 3772t$$

The second approach we use is to calculate the total mass with the software Siemens. Imposing the same density, homogeneous in the deck, it is possible to compute both volumes and masses. The result is shown in the following data, the complete report is in appendix A.

==

Misurazione proprietà di massa

Visualizzati valori proprietà di massa

Volume = 1246 m^3

Area = 1797m^2

Massa = 3116492 kg

Peso = 30562349N

Raggio di inerzia = 12.62 m

Baricentro = 8.368 8.899 42.468 m

==

Proprietà di massa dettagliate

Analisi calcolata con una precisione di 0.990000000

Unità di misura informazioni kg - m

Densità = 2500

Volume = 1246

Area = 1797

Massa = 3116492

It is possible to see, for the computed mass, that is similar to that calculated with the geometrical method, it defers for 500 tons, but still is more than 3000tons. The program calculated a more precise volume and so a more precise masse. We developed many calculations with different densities, these are illustrated in appendix A.

Comparing the calculated data (from geometry analysis and from Siemens) with that coming from the measurement we obtain a difference of thousands of tons. The measured data is about 1200t. To fit and understand this difference the only one solution is that the deck is not designed with a full section, instead is made by many empty coffers. This assumption is confirmed from the designers.

2.2) Underground works characterization

The deck impact on the ground and the consequent vibration might compromise the integrity of the buried structures under the bridge. These buried structures are all the services pipeline. For instance, cables for the electricity, pipeline for drinkable water, pipelines for gas transport and so on and so forth. Those lines go through ground under the bridge. Their path follows virtually the path of the bridge. They are buried 2 meters deep under the free surface.

Their structure is complex. They are not easily pipeline buried in the soil. The pipeline or the cable lies in a sand or gravel bed that surround it. All inside a reinforced concreate box. It is possible to approximate them as follow:

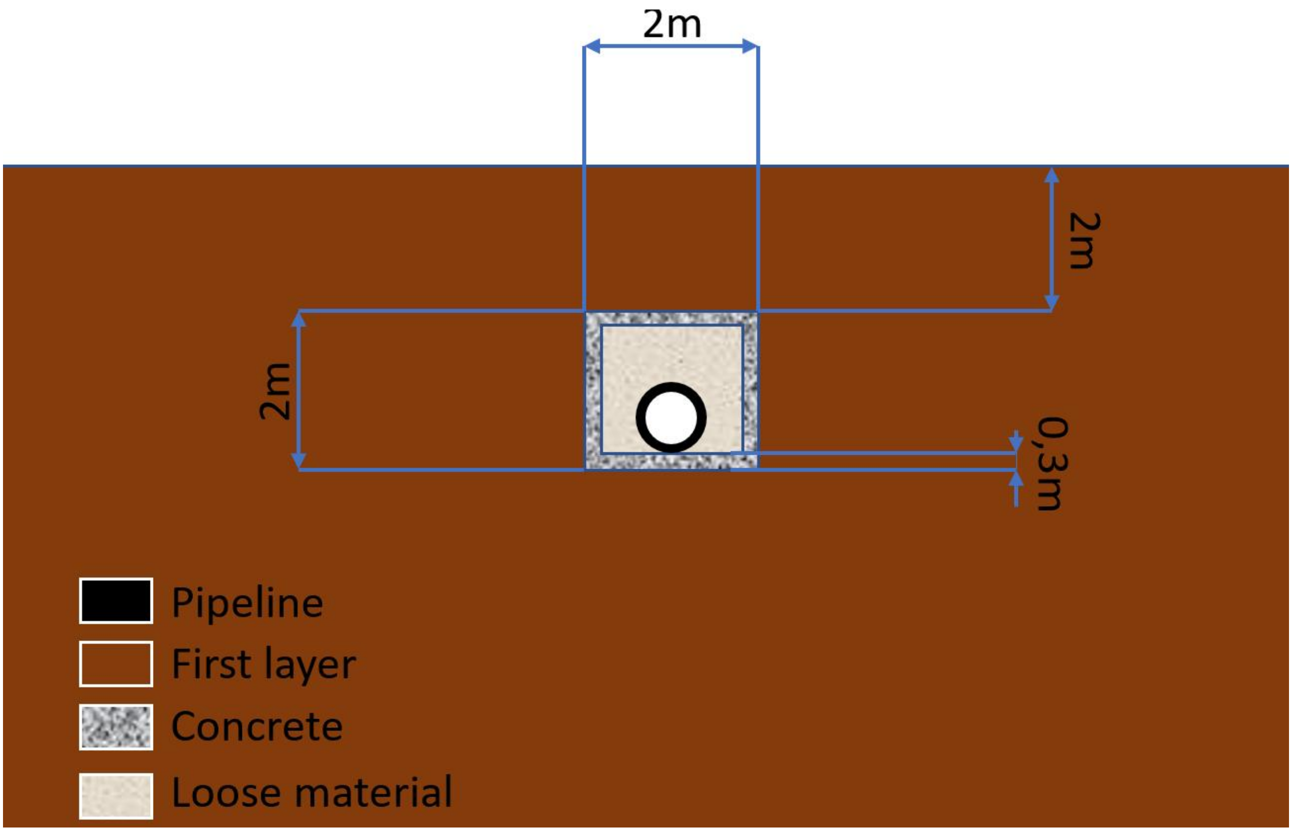


Figure 4It represent a generic sketch of a buried pipeline. We are going to model our problem exactly following this sketch. The pipeline inside a concreate box buried at 2 meters depth.

From figure 4 it is possible to understand how we want to model the geometry of the underground problem. The pipeline in into a reinforced concreate box. The box is a two meters depth and is 30cm thick for all its perimeter. The properties of the concreate are retrieved from table1. We suppose an elastic modulus equal to 25GPa. The surrounding ground has the characteristics of the first layer and will be described in the following paragraph.

Our goal is to understand how huge the normal stresses are. They are due to the impact of the deck, and act on the upper concreate surface. It is important also understand the distribution of the stresses all around the box.

The first approach we use to roughly approach to the problem is the Kirsh law. It is valid just for cylindrical geometries and, most important, for rocks. The roughly analytical approach will be followed by a numerical modelling. Firstly, with the structural mechanics tool of matlab, on which everything is elastic, to arrive at the viscoplastoelastic study with Flack.

2.3) Geophysical characterization

The model is based on the parametrization of the geophysical and geotechnical data of the ground below the pillar. Those data come from seismic assay on the surface and in down hole. The simplified model of the ground is design starting from the compression and shear wave velocities found with the aforementioned assays. The data are collected in the table 2.

Table 2: geophysical parameter for the propagation seismic way modelling.

Id	**Description**	**Depth [m]**	**Pwave velocity Vp [m/s]**	**Swave velocity. Vs [m/s]**	**Quality factor*(P) Qp [-]**	**Quality factor (S) Qs [-]**	**Density [kg/m^3]**
1	Debris cover of an anthropic nature and coarse alluvial deposits	0 -10	800	300	10	10	1800
2	Low consolidated sands and gravels	10-20	2000	300	50	50	1800
3	Consolidated sands and gravels	20 – 30	2500	400	200	200	1900
4	Deposit flyshoidi	30	2800	800	200	200	2000

Quality factors come from literature they are not experimentally obtained.

To fully complete the ground characterization, it is necessary calculate the elastic properties of the ground, so the E, G and ν. Their formulations are listed below:

$$\nu = \frac{V_s}{V_p}$$

$$G = {V_s}^2 * \rho$$

$$E = G * (2 * (1 + \nu))$$

By the three equations listed above it is possible to calculate the modules for each stratum, they are listed in the table 3.

Table 3: Values of modules for the respective layers

ν	**G[Pa]**	**E=[Pa]**
0,375	162E+06	450E+06
0,15	162E+06	389E+06
0,16	304E+06	705E+06
0,285	1280E+06	3289E+06

2.4) Analysis of the impact

For the calculation of impact pressure, it is necessary to define the physic of the problem. Our data are:

- m=1200t
- H=45m
- A=630m^2

Where m is the mass of the deck, H is the height of the fall and A is the impact area, simply found multiplying the length and the width of the deck.

The velocity of impact is calculated considering the deck as a grave falling at the ground. The air attrition has been neglected.

$$v = \sqrt{2gH} = \sqrt{2 * 9.81\,m/s^2 * 45m} = 30\,m/s$$

- v=30m/s

this is the velocity of the deck at the moment of the impact with the ground. We considered negligible the attrition of the air to be more conservative with the calculations.

The force applied on the ground is supposed to be a pulse force. For definition the pulse force is applied for a given interval of time. The time of pulse must be little enough to approximate the time of impact of the deck.

$$J = \int_{t_i}^{t_f} F(t)\,dt = \int_{t_i}^{t_f} \frac{dp}{dt}\,dt = \int_{t_i}^{t_f} dp = m(v_i - v_f)$$

Where J is the pulse [N*s], t_i and t_f are the initial and final time of impact [s], F(t) is the applied force [N], dp is the differential of the linear momentum [kg*m/s], v_i and v_f are the initial and final velocity of the deck during the considered time interval [m/s]. The initial velocity is that calculated before v_i=30m/s. Some considerations have to be done for final velocity and interval of time. The time interval is that characteristic of the impact and it is a function of the ground typology. Considering an unconsolidated soil, it can vary between 0,05s up to 0.2s. The initial time is equal to 0s. From the equation written above we impose the conditions of t_i=0s and $t_f \in$(0.05s-0.2s). We obtain and interval of forces between 180kN and 720kN. The impact pressure is calculated considering the net contribution of the weight force plus the impact forces:

$$P_{impact} = \left(\frac{mV_i}{dT} + mg\right)/A$$

It is obtained, for an impact time equal to 0,1s an impact pressure of 585kPa.

To evaluate the forces which play a significant role on the physics of the problem it is necessary to evaluate the reaction force of the ground. The reaction force gives an idea of the real impact force of the deck on the ground surface. To do it is needed the deceleration of the deck and thus the static displacement of the ground. The static ground displacement, for a granular soil, could be retrieved in more than one way. The most suitable methods are that of Boussinesq and that of Westergaard. It is also useful the Burland's method. The ground with the properties collected in table 2 and the impact area considered, give us the possibility to calculate the displacement with the following relation:

$$\frac{w}{q} = (6.53 + 0.47B)$$

Where w is the displacement, measured in mm, q is the pressure acting on the given surface of impact, measured in Pa and B is the width of the impact area, expressed in m. Substituting our values we obtain a displacement almost equal to 125mm.

Once calculated the static displacement it is possible to understand how the ground reacts to the impact force of the deck. The impact forces are linked to the reaction of the ground hence with the stopping time of the deck. The link among them is the stiffness characteristic of the ground. The velocity of the impact decreases exponentially with the time, as follows:

$$v(t) = v_i\left(e^{(-k\,t)}\right)$$

Where k is a parameter dependent on the ground deformability. For unconsolidated ground it increases with the compaction factor as is possible to see in table 4.

Table 4 estimation of the impact time, deformation and impact pressure of different grounds. The considered mass is that of the deck, 1200t from an height equal to 45m

	K coefficient	Impact time [s]	Deformation [m]	Impact pressure [kPa]
Dense soil	150	0.05	0.2	1150
Less dense soil	50	0.10	0.6	600
Lose meterial	25	0.20	1.2	300

The space covered by the deck meanwhile is impacting and deforming the ground is calculated by integrating the previous equation:

$$d(t) = \int v(t)\,dt = v_i \int \left(e^{(-k\,t)}\right)\,dt = \frac{v_i}{k}\left(e^{(-k\,t)}\right)$$

Our hypothesis is a 1200t deck with an impact velocity of 30m/s from a height of 44m. With those hypotheses it is possible to estimate the slowing down velocity of the deck meanwhile the grounding is deformed. The results are plotted in the figures 5 and 6 below. From those pictures it is also possible to retrieve the impact pressure for different values of k (those reported in table 4). This is represented in picture 7.

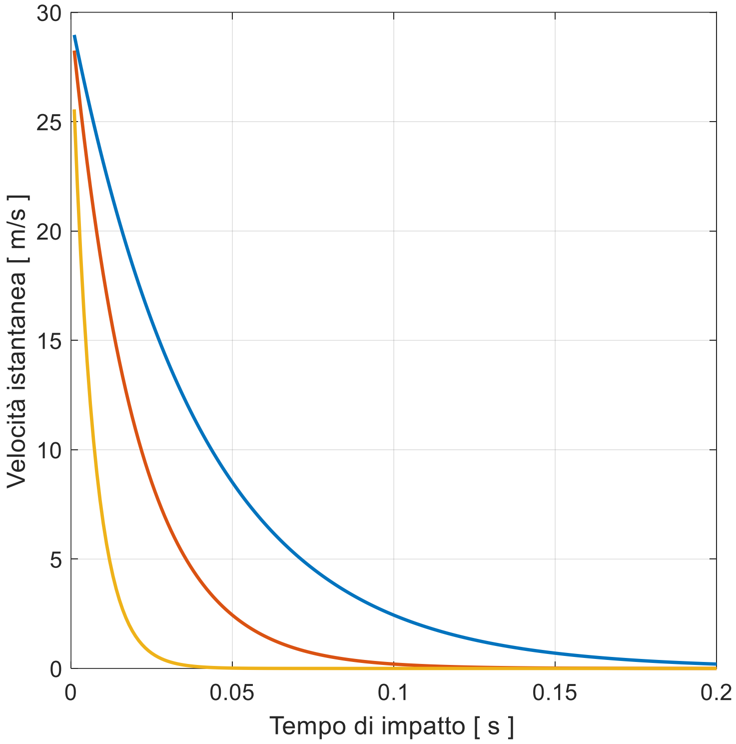


Figure 5 Instantaneous velocity of the deck during the impact phase on the ground. The different curves represent grounds with different deformability. Blue k=25, red k=50, orange k=150.

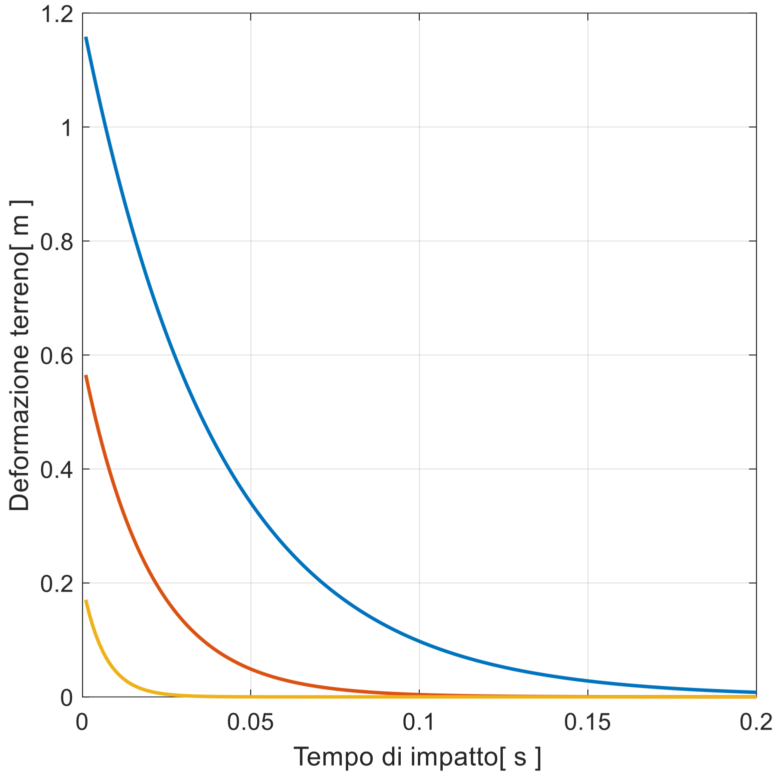


Figure 6 Stopping time of the deck during the impact phase on the ground. The different curves represent grounds with different deformability. Blue k=25, red k=50, orange k=150.

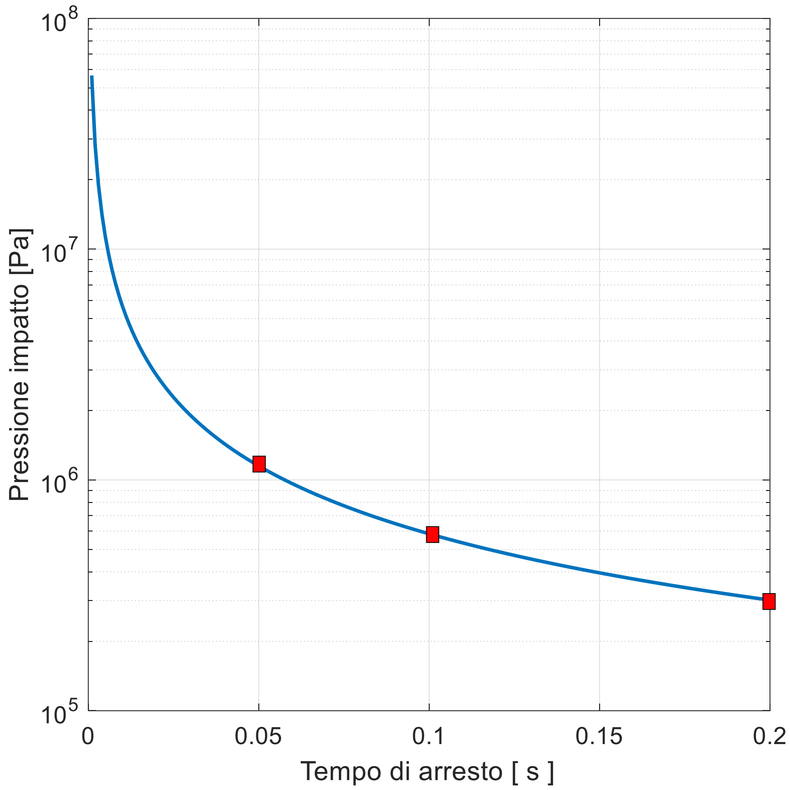


Figure 7 Impact pressure function of the stopping time of the deck. Red symbols represent the value of k=25, k=50 and k=150

One important aspect for both the models we must develop, analytical and numerical ones, is the time over which the impact lies. To estimate this time, we need to understand how the deceleration of the deck is once impacted with the ground. Hence, we need to discover, from the literature, how the ground is squeezed by an impact mass.

In general, to estimate how the ground is compacted, when subjected to an impact of a massive body, is used the theory of the Heavy Tamping. The heavy tamping is a technique developed around the70th, developed by Menard.

The method consists in the application of a sequence of heavy tamping over a ground surface. In general, the tamping is generated by a concreate block or a steel slab but also by steel boxes filled with concreate. The masses of the tamping in general are between 5 and 200 tons, actually the most used are around 15-20t. They fall down from height that can reach 40 meters, and minimum are 7 meters.

It is possible to resume the application scheme as in figure 8:

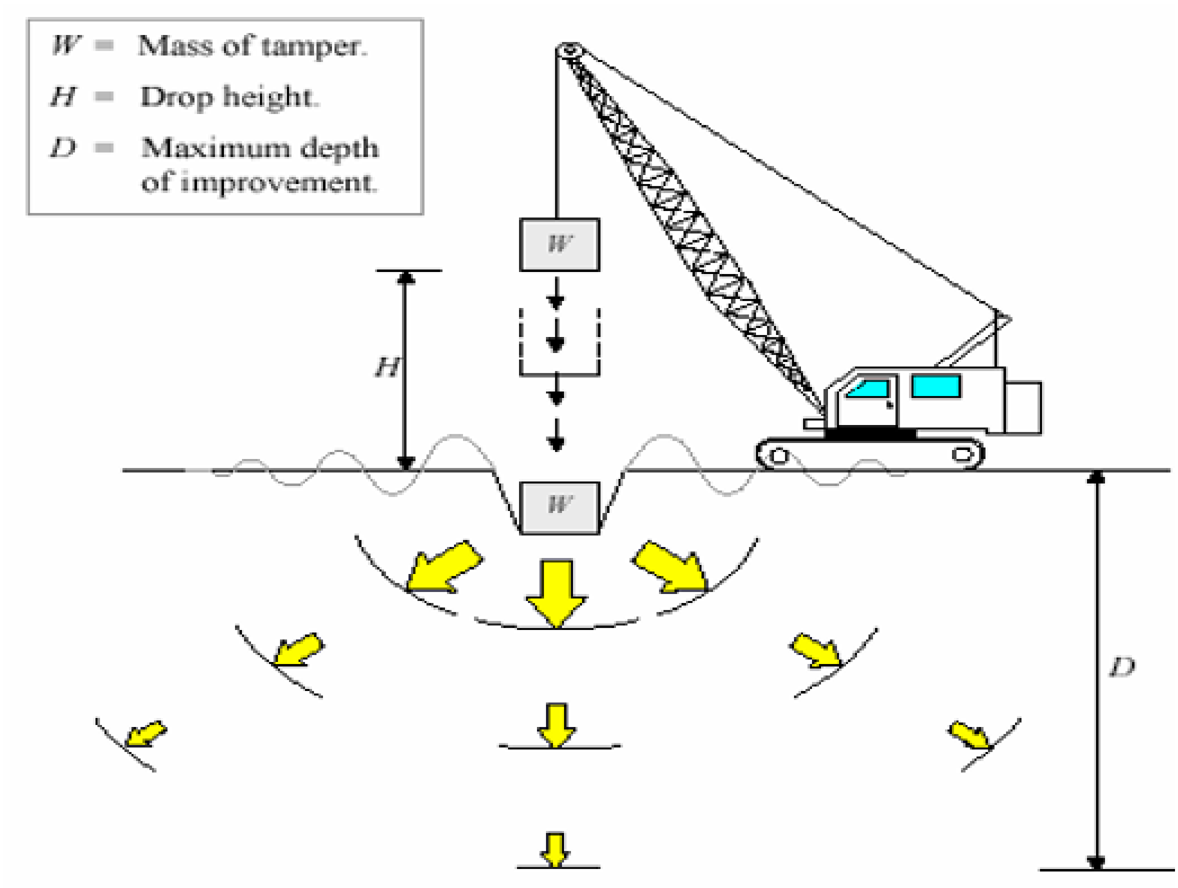


Figure 8 Scheme which represent the heavy tamping technique. The mass (W) is lifted by a crane at the desired height (H). the effect on the ground is the induced vibration but also a vertical permanent displacement (D)

Our interest is to understand the magnitude of the displacement, this in order to calculate a generic time of impact. The formula to evaluate the displacement is the following:

$$D_{max} = n\sqrt{W \cdot H}$$

Where W is the mass of the tamper, H is the drop height and n is a coefficient which consider the characteristics of the soil and the variability of the equipment. The n coefficient is dimensionless and can vary, typically, from 0.3 for very hard soils and 0.6 for very soft soils. The first layer of our geometry is a medium soft soil. Its characteristics are listed in table 2 and 3.

We may suppose the n coefficient equal to 0.4. To calculate the displacement we have to suppose, also, the height of drop and the weight. We must suppose it because our dimensions are bigger then the maximum used for this king of tests. To understand the order of magnitude of the impact time we may do many simulations with different weights and heights.

We suppose 2 weights, 12t and 120t, and five heights, 10m, 20, 30, 40 and 45m.

The results are shown in table 5:

Table 5: shows the displacement obtained with the heavy tamping technique for different masses and different heights.

H[m]	W[t]	D[m]
10	12	3,28
20	12	4,64
30	12	5,69
40	12	6,57
45	12	6,97
10	120	10,39
20	120	14,69
30	120	18
40	120	20,78
45	120	22,04

To calculate the corresponding stopping time, we must use the kinematic equations of the uniformly accelerated linear motions.

$$\begin{cases} v^2 = {v_0}^2 + a(x - x_0) \\ v = v_0 + at \end{cases}$$

Where v is the final velocity, hence, equal to 0, v_0 is the initial velocity equal to 30m\s, a is the deceleration of the body, t is the time needed for stopping, x-x_0 is the D_{max}. The displacement found in table 5 are used in the first equation in order to find the acceleration, then is possible to use the second equation to retrieve the time. The results are collected in the table 6:

Table 6: in the first column is reported the vertical displacement due to the impact of the mass, in the second column is reported di deceleration of the mass and in the third column is reported the time needed to stop.

D[m]	a[m\s^2]	t[s]
3,29	136,93	0,22
4,65	96,82	0,31
5,69	79,06	0,38
6,57	68,47	0,44
6,97	64,55	0,46

10,39	43,30	0,69
14,70	30,62	0,98
18,00	25,00	1,20
20,78	21,65	1,39
22,05	20,41	1,47

From table 6 we may appreciate that for same height of 45m the stopping time is between 0.22s and 1.47 seconds. The height of 45 meters is equal to that of our problem. There is still a great difference with the considered masses, our is about 1200t while the maximum for tamping is about 200t. Another consideration which could be done is try to correlate the weight per unit area. To understand the area of application of the tamper is possible to look at figure 9:

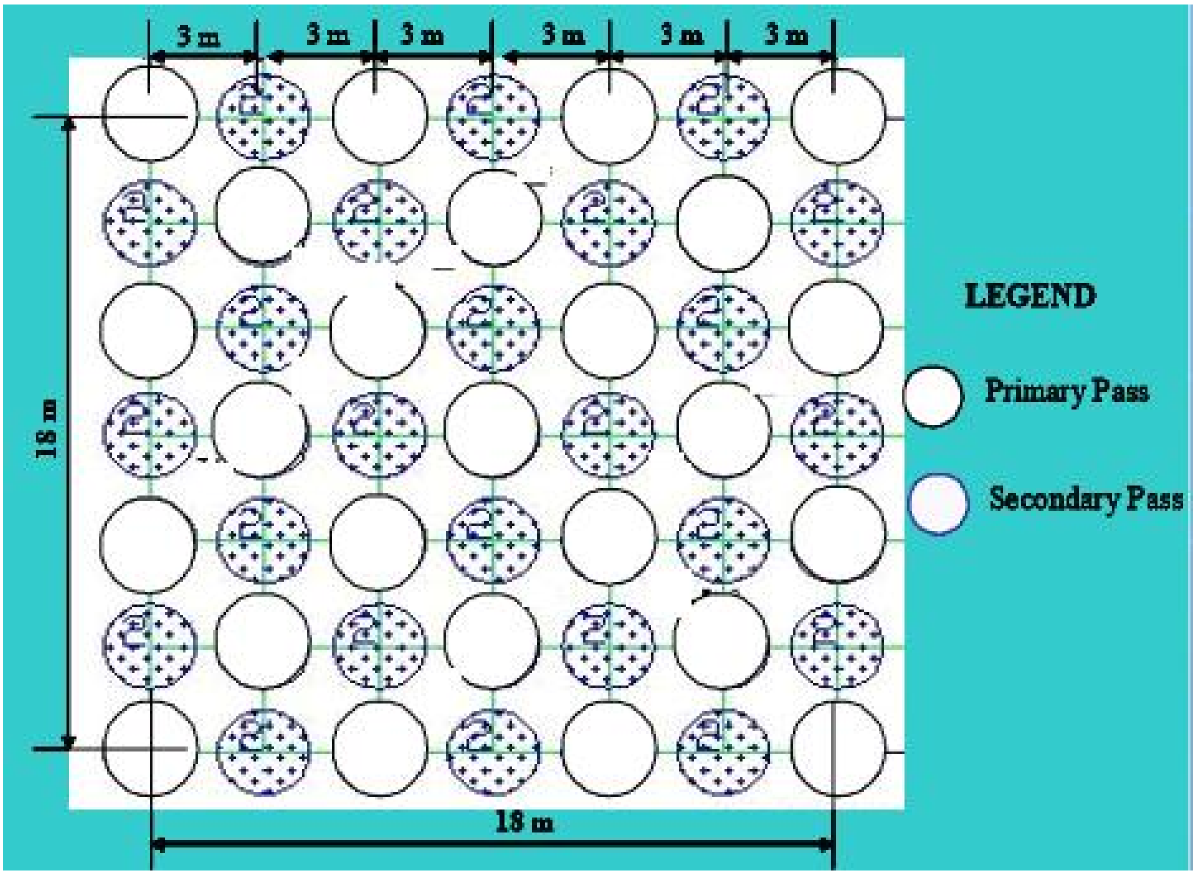


Figure 9 This is the schematic representation of the heavy tamping technique. It is possible to see that it is made by many phases, It is interesting to understand that the radius of the tamper is 1,5meters, and so its area is about 7m^2.

Our body has 1200t distributed over and area of 675m^2, hence it has a 1.9t/m^2. The tamper, generally, are cylinder of 7m^2 of surface. That means the tamper considered in table 5 and 6 have a weight for unit area between 1.7kg\m^2 and 171kg\m^2.

Hence the interesting datum is the green one following:

H[m]	W[t]	D[m]	a[m/s^2]	t[s]	W/S [kg\m^2]
10	12	3,29	136,93	0,22	1,71
20	12	4,65	96,82	0,31	1,71
30	12	5,69	79,06	0,38	1,71
40	12	6,57	68,47	0,44	1,71
45	12	6,97	64,55	0,46	1,71

With those assumptions we can say that the deck, during the impact, will stop its run in 0,46 seconds. This is true if the ground is unconsolidated, our ground could be considered as consolidated. In our case it is possible to estimate a vertical displacement of some centimeters, that means our time are, at least one or two order of magnitude lower than seconds. We suppose 0.05seconds of stopping time.

It is important to notice that, thanks to the heavy tamping method, we can also understand the amount of the energy converted in vibrations during the deck impaction. The heavy tamping theory suggests that dynamic compaction generates surface waves between 3 and 12Hz. Also, it is possible to say that the energy of the vibrations is subdivided as follows in figure 10:

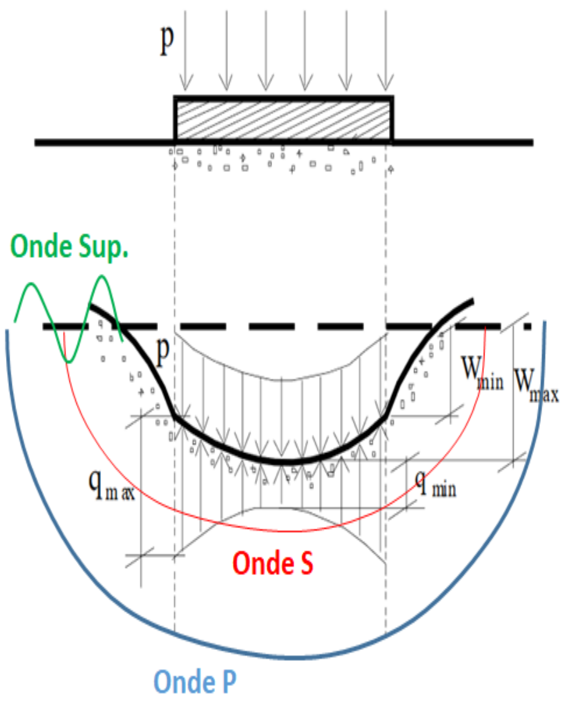


Figure 10 Represent the subdivision of the impact energy in different form of waves, in blue the compression waves, in red the shear waves and in green the surface waves.

The partition of impact energy in the three different phenomena is calculated combining the following equations:

$$u_R = \frac{P^*}{2\pi GR} \frac{\cos\theta\,(\mu^2 - 2\sin^2\theta)}{F_0(\sin\theta)} \cos(\omega t - k_1 R)$$

$$u_\theta = \frac{i\mu^3 P^*}{2\pi GR} \frac{\sin 2\theta\,(\mu^2 \sin^2\theta - 1)^{1/2}}{F_0(\mu\sin\theta)} \sin(\omega t - k_2 R)$$

$$\bar{u}_r = \frac{P^*}{G}\left(\frac{k^3}{2\pi r}\right)^{1/2} F_r(\nu)\sin(\omega t - k_3 r - \pi/4)$$

$$\bar{u}_z = \frac{P^*}{G}\left(\frac{k^3}{2\pi r}\right)^{1/2} F_z(\nu)\cos(\omega t - k_3 r - \pi/4)$$

Where: u_r and u_θ are respectively the radial and the transverse displacements, P* is defined ad P(t)/cos(ωt) and is the impulsive load, θ equal to 1/cos(z/R), k_1 is equal to ω/c_1, k_2 is equal to ω/c_2, μ is equal to $c_1/c_2=[2(1-\nu)/(1-2\nu)]^{1/2}$, $F_0(\zeta)=(2\zeta^2-\mu^2)-4\zeta^2(\zeta^2-\mu^2)(\zeta^2-1)$, where ν is the Poisson's ratio and ζ is the dumping ratio, k_3 is equal to ω/c_3, $F_r(\nu)$ and $F_z(\nu)$ are functions of Poisson's ratio.

The result is:

- Compression 7%
- Shear 26%
- Rayleigh 67% (Miller & Pursey, 1955)

Rayleigh waves stand for the 67% of the total vibration energy. Furthermore, they become predominantly over other waves at comparatively small distances, from the production source. Building foundations and other civil constructions are particularly subjected by Rayleigh waves.

3) Modelling

In this chapter we will model the problem in two principal ways.

1. Analytical model
2. Numerical model

The former is the analytical resolution of the differential equation which describes better the physics of the impact, then an easy analytical approach to the calculation of stresses over the underground works by the resolution of Kirsh problem. The latter is the numerical modelling performed with different tool, Matlab and Flack for both, the impact and the stresses on the buried structures.

In the first part we search a physical model which fits with our case. We will decide between different viscoelastic models, described by second order differential equation forced and dumped. To solve those equations, we need to understand and hypothesize the boundary conditions and the Cauchy problem, which means the initial condition at which the model is subjected. For what concerns the modelling of the buried pipeline the approach will be more qualitative, to have just an idea of the stresses in place.

With the second approach we try to approximate the problem with the numerical simulation, it means we create a model with a geometry similar to that of our problem. This geometry will be filled by the geophysical properties of the model. Once done it will be possible to impose the boundary condition, similarly to it done for the analytical solution. The first approach with the numerical model will be done with a media completely elastic, to understand well the physics. Than we will move through a viscoelastic modelling. Then, to model the behavior of the stresses on the pipeline we will solve a numerical model in completely elastic material for moving in a second time in a viscoelastic model that will describe the problem in a more realistic way.

3.1)Analytical model

The model is a forced vibration with dumping problem also called viscoelasticity problem. To solve analytically the problem, we need a physical-mathematical approximation. The models that better approach to the problem are two. The Maxwell model and the Kevin-Voigt. Those two models are:

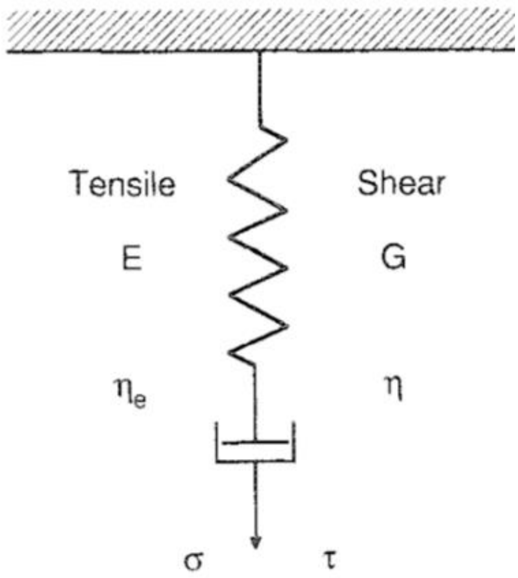


Figure 11 Maxwell model, it is a series of a spring and a dashpot, in the figure are reported tensile stress σ, the shear stress τ, the elastic modulus E, the Shear modulus G, the constant of elasticity and shear of the dashpot η_e and η.

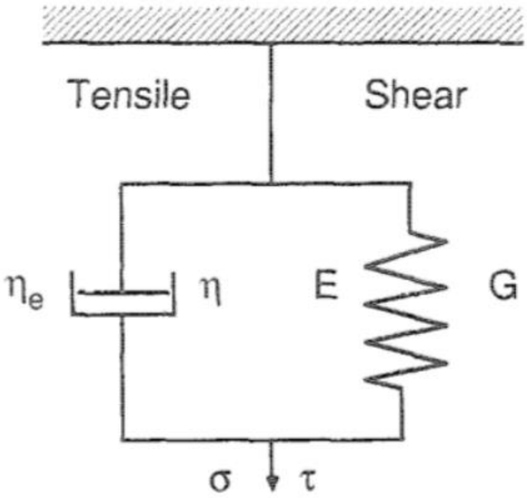


Figure 12 Kevin-Voigt model, it is a parallel of a spring and a dashpot, in the figure are reported tensile stress σ, the shear stress τ, the elastic modulus E, the Shear modulus G, the constant of elasticity and shear of the dashpot η_e and η.

Our choice is the Kevin-Voigt model. For convenience we split in two the dashpot, obtaining this scheme:

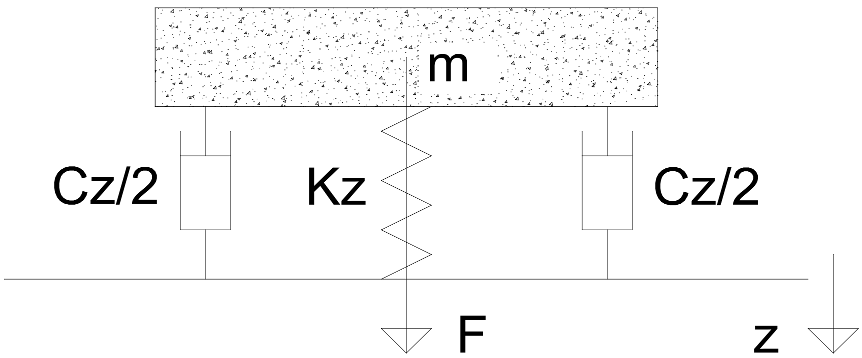


Figure 13Sping-dashspot system used to simulate the vertical displacement at the interface between the ground and the deck. The deck mass is m, the dumping coefficient is cz, the stiffness of the spring is Kz, z is the vertical displacement and F is the impact force.

The dashspots have half of the total dumping ratio each one. Their goal is to simulate the dissipation due to the resistance force proportional to the vibration velocity.

The dynamic equilibrium is given by the following differential equation:

$$m\ddot{z} + c_z\dot{z} + K_z z = F(t)$$

Where m is the mass of the deck, c_z is the dumping, K_z is the stiffness, $z, \dot{z}, \ddot{z}$ are respectively the vertical displacement, the vertical velocity and the vertical acceleration and F(t) is the impact force.

K_z induces the stiffness characteristic at the interface between deck and ground. Its value depends on geometrical characteristic of the deck (length), by the shear modulus of the ground (G) and by the Poisson coefficient (ν), all in accordance with the following:

$$K_z = \frac{S_z\, L\, G}{1 - \nu}$$

Where S_z is an semi-empiric coefficient equal to 0.8. It comes from Whitman, Richart, Dobry, Gazetas studies. The K_z is measured in N/m. L is the impact length, so is equal to the length of the deck, the shear modulus and the Poisson's coefficient are those relative to the first layer.

Hence,

$$K_z = \frac{0.8 \cdot 42m \cdot\ 162 \cdot 10^6 Pa}{1 - 0.375} = 8{,}7 \cdot 10^9\, {}^{N}/_{m}$$

This is the stiffness constant for the spring associated to the first layer.

The other value of our interest is c_z. It is the dumping coefficient. When it is critical is calculates with:

$$c_z = n\sqrt{K_z\, m}$$

Where, Kz is the stiffness of the material, m is the impact mass and n is a coefficient which depends from the ground characteristic and the impact masses. From empirical tests we found that a value around 0.2 could be a good approximation for the case of study. Thus:

$$c_z = 0.2\sqrt{K_z\, m}$$

Now it is possible to define the boundary conditions. We have two hypotheses; the former is the following:

$$\begin{cases} F(t) = 0N & t = 0 \\ v = 30m/s & t = 0 \\ z = 0m & t = 0 \end{cases}$$

In this case we have a differential equation which is homogeneous. To solve it is enough to solve the associated equation, obtaining:

$$m\ddot{z} + c_z\dot{z}\ +\ K_z z = F(t)$$

Substituting the values

$$1200x10^3\ddot{z} + 13.14x10^6\dot{z}\ +\ 8{,}7x10^9 z = 0$$

Solving the associated equation

$$1200x10^3\lambda^2 + 13.14x10^6\lambda + 8.7x10^9 = 0$$

The solutions are:

$$\lambda_{1,2} = \frac{-13.14x10^6 \mp \sqrt{\left(13.14x10^6\right)^2 - 17280x10^{12}}}{2.4x10^6}$$

$$\begin{cases} \lambda_1 = -5.475 - 84{,}97i \\ \lambda_2 = -5.475 + 84{,}97i \end{cases}$$

The real part of the solution is the exponent of the exponential part and the imaginary part is the argument of the trigonometric part of the homogeneous integral.

$$z = e^{-5.475t}(C_1 cos(84{,}97) + C_2 sen(84{,}5))$$

Imposing the boundary conditions, we find:

$$\begin{cases} C_1 = 0 \\ C_2 = 0{,}3539 \end{cases}$$

Hence

$$z = e^{-5.475t}(0.3539 sen(84{,}97))$$

Then the plot of the displacement and the velocity is shown in figure 14 and 15:

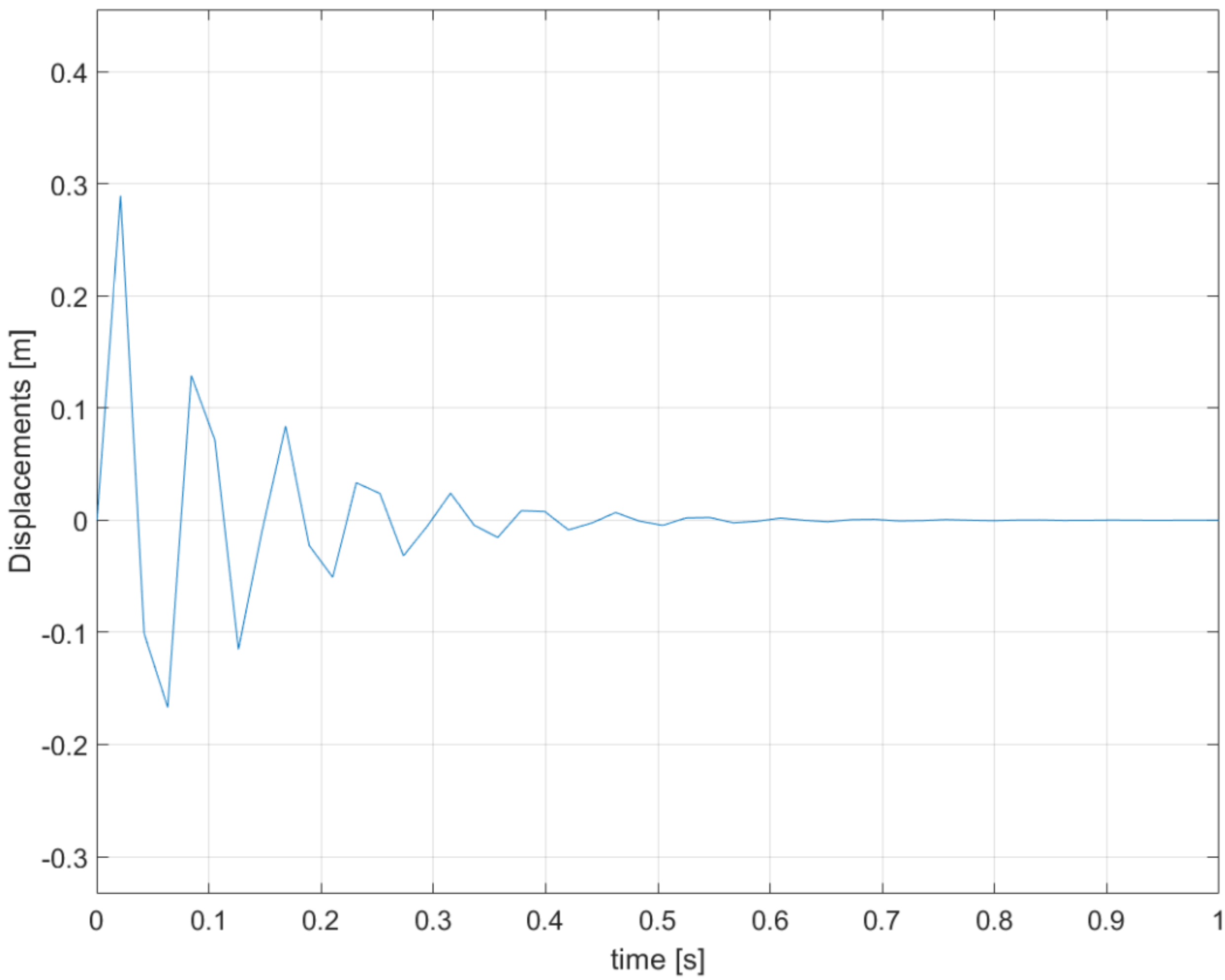


Figure 14The plot represents the solution of our differential equation. In ordinates are reported the displacement in abscissae are reported the time. The maximum displacement is about 0,5 m in the early time.

From the plot in figure 14 it is possible to see how, at impact time, the displacement due to the deck arrive is. The maximum displacement is around 0,25 cm. From the plot it is also possible to see that the displacement seems to be not permanent. After some oscillations it returns to 0. From the physical point of view, it does not make sense. When the ground is hit by the deck the deformation is permanent. That means the displacement of the ground cannot return to 0 once been higher than 0.

In the plot in figure 11 we can have an idea to how high are the vibrations in this case. From the plot is not possible to see it but the maximum value of the vibrations is about 30m/s at the impact. Those vibration will be subdivided in P, S and Rayleigh waves. Either will decrease with the distance.

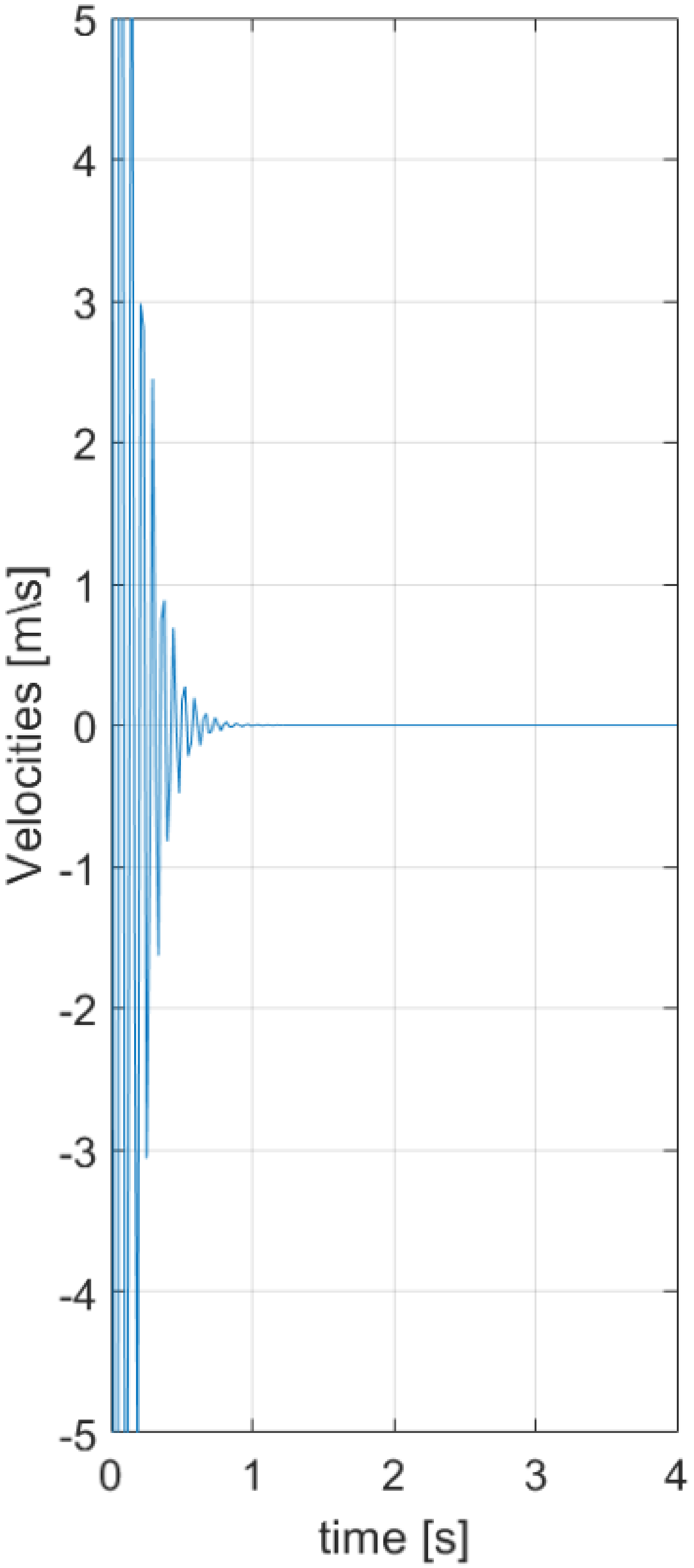


Figure 15This plot represents the first derivative of our solution. This time in ordinate we have the velocity. The highest value of the velocity in the plot is almost 30m/s.

We may say that the hypothesis done for solving the problem present some incongruency. Firstly, it is possible to see from the plot in figure 14 that the displacement returns to 0 m with increasing the time. This is impossible because the ground is permanently deformed. The second incongruency is the velocity of the vibrations, they are too high.

The latter hypothesis we thought is the following boundary conditions:

$$\begin{cases} F(t) = 180000000N & t = 0 \\ v = 0m/s & t = 0 \\ z = 0m & t = 0 \end{cases}$$

These boundary conditions mean the force applied to the ground is the force of the falling mass, the velocity of the system at the collision time (t=0s) is equal to 0 the same value is supposed for the displacement. Our differential equation has the same solution for the homogeneous part. We must solve the particular integral of the following differential equation:

$$m\ddot{z} + c_z\dot{z} + K_z z = F(t)$$

$$1200x10^3\ddot{z} + 13.14x10^6\dot{z} + 8{,}7x10^9 z = 180000000$$

Our forcing functions is simply a constant, for the similitude method we must choose a constant too to solve the particular integral.

So:

$$z_p = A$$

Where A is a constant, now we substitute it into the differential equation:

$$m\ddot{z} + c_z\dot{z} + K_z z = F(t)$$

$$0 + 0 + \ 8{,}7 \cdot 10^9 = 180000000$$

$$z_p = 0.0207$$

Since the solution of the differential equation is the sum of the homogeneous and the particular integral, we obtain:

$$z = e^{-5.475t}(C_1 cos(84{,}97) + C_2 sen(84{,}97)) + 0.0207$$

Now is possible to find the new constant, if enough to impose the new boundary conditions:

$$\begin{cases} C_1 = -0.0207 \\ C_2 = -0.0021 \end{cases}$$

The general solution will be:

$$z = e^{-5.475t}(-0.0207cos(54.5) - 0.0201sen(54.5)) + 0.0207$$

The results are shown in figure 16 and 17. From the plot in figure 16, which represent the displacement versus time, it is already possible to see a big difference with the plot shown in figure 14. The plot in figure 14 shows the tendency of the displacement to return to 0. This was evidently a mistake done in the boundary conditions. With the new boundary conditions, we reached this new result, it is more realistic. The displacement in this case is lower than before but presents a permanent deformation. The upper value reached by the displacement is about 0.03m. Then 0.3s later the oscillation stops and tends to a permanent displacement value equal to 0.025m This means the ground has a permanent deformation, due to the impaction of the deck, equal to 2,5 cm.

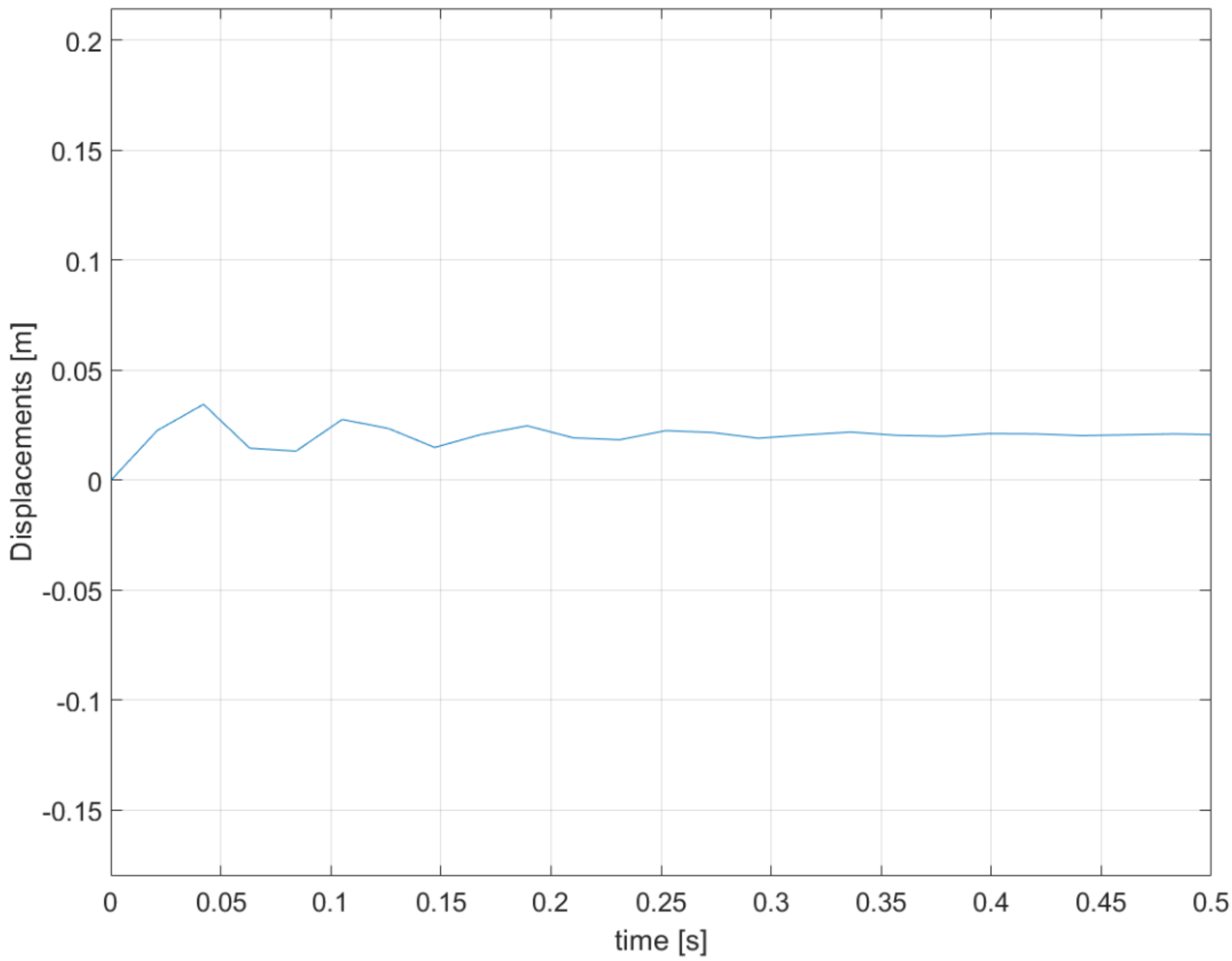


Figure 16 The plot represents the solution of our differential equation. In ordinates are reported the displacement in abscissae are reported the time. The maximum displacement is about 0,03 m in the early time.

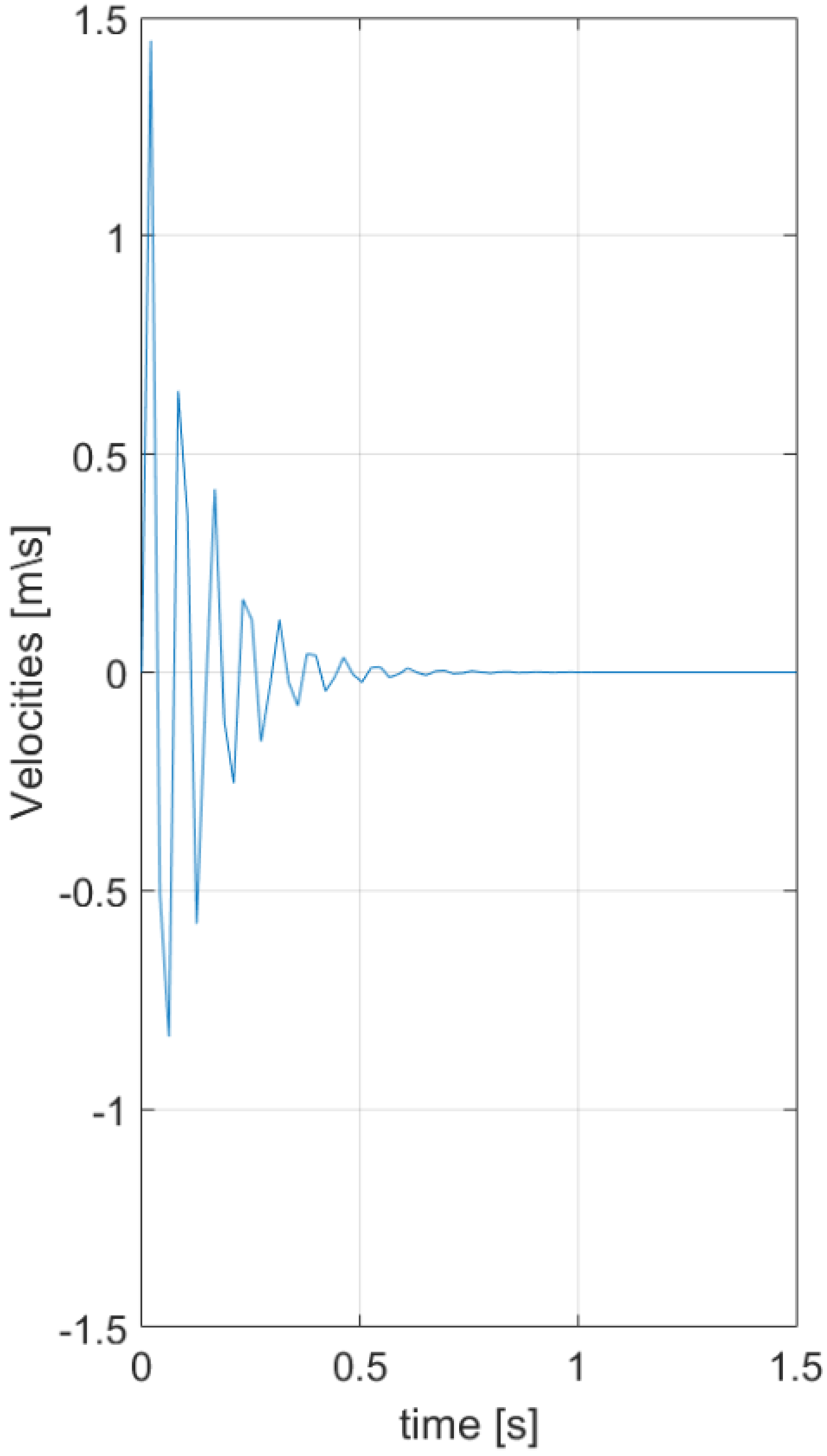


Figure 17This plot represents the first derivative of our solution. This time in ordinate we have the velocity. The highest value of the velocity in the plot is almost 1.5m/s.

The second result, thus, the plot in figure 17 presents also some differences compared to the previous one, thus that in figure 15. The vibration picks this time is lower from before. It is almost 5 times lower. In figure 17 the velocity, at the impact time, reaches 1,5m/s. This time the vibration goes down in the half of the time. In plot 15 was necessary 1 second, now, in plot 17 just 0,5 seconds. Those two plots, 16 and 15, give a more reliable results than the previous model. The results give a more accurate approximation of the fact. For the other cases, for example the cases with the presence of a pillow of sand will be discussed with the same boundary conditions. One more thing can be said. It is possible that the mathematical form of the force, so the forcing function, could not be a constant but could be a sinusoid or co-sinusoid. We will analyze this problem later.

Further from the impact zone is needed an analysis of the propagation of the waves. Considering the ground, a homogeneous media we might assume the motion equation as follow:

$$\rho\ddot{\boldsymbol{u}} = (\lambda + 2\mu)\nabla^2\boldsymbol{u} - \mu\nabla \times \nabla \times \boldsymbol{u}$$

Where **u** is the particle displacement vector, it is induced by the passage of the waves, λ and μ are the Lamè constant of the ground. This equation is used to estimate the propagation of vibrations for different distances from the impact zone, so, from the source of the vibrations.

The elastic energy decreases during the propagation, this is due to various mechanisms, they are:

- Dumping due to the viscoelastic properties of the material
- Geometrical dispersion, the energie of surficial waves decreases with the square root of the distance, while that of shear waves and pressure waves decreases with the distance
- Scattering dissipation, due to the heterogeneities of the ground

The simplified solution of the previous equation is this:

$$u(r,t) \; = \; \frac{u_0}{0.5r} \; [\, exp(-\pi \; f \; r/Q \; c) \,]$$

This is valid under material with viscoelastic behavior. u(r,t) represent the amplitude of the vibration at the distance equal to r from the impact zone, u_0 is the reference value, f is the frequency of the oscillation, Q is the quality factor and c is the waves velocities.

For volumetric waves, hence P and S waves the geometrical attenuation is proportional to 1/r. Structures have their tolerance limits regarding vibrations. In general, we might say that reinforced structures can sustain 50mm\s for 4Hz waves. Under reinforced structures as residential ones may sustain 15-20mm\s at 4-15Hz and 20-50mm\s for 15-40Hz.

In Italy we have a strictly legislations for this field. The regulation in use is the ISO 4866. It contains the rules and the guidelines principally adopted as reference point. Also, swiss and German regulation are adopted. Some of the parameters considered by the ISO 4866 are the frequencies range, the amplitude of the oscillations, the timing of the event if transient or continuous, the number of the events, the exposure time, the ground influence and at last the condition of the structure which we want to preserve.

From the ISO 4866 a more specific law might be retrieved, it is the UNI 9916 which gives us the reference values for low time vibrations (which is our case). Those limits are listed in the tables 1 into APPENDIX B.

Also, for the underground structure we must look to the normative. For this case is taken into consideration the swiss normative. Precisely the SN640°312 of the 1992. It stabilizes the limits of the vibration velocity in terms of number of events that interact with the underground structures. The reference value is not anymore, the (p.c.p.v) but the "peak particle velocity", (p.p.v.) which mans the pick value of the vibration over a given time.

Those thresholds are listed in table 2 presents in APPENDIX B.

The category which is of our interest is the second for table7 and from table8 the class C, It is possible to see that the upper limit for every frequency is 15mm\s. It is evidently that our case of study, until now cannot respect the law. If we look to the figure 17 it is possible to see that the peak velocity at the impact presents 1,5m\s. If the source of vibration is at 1,5m\s is impossible, using the 1\r law, that the near and medium distance buildings will be preserved.

It is possible to see in figure 18 how the surface and shear waves propagate from the source in the surrounding ground. For distance surface waves we can notice that a great distance, over 100m we are hugely above the low limits.

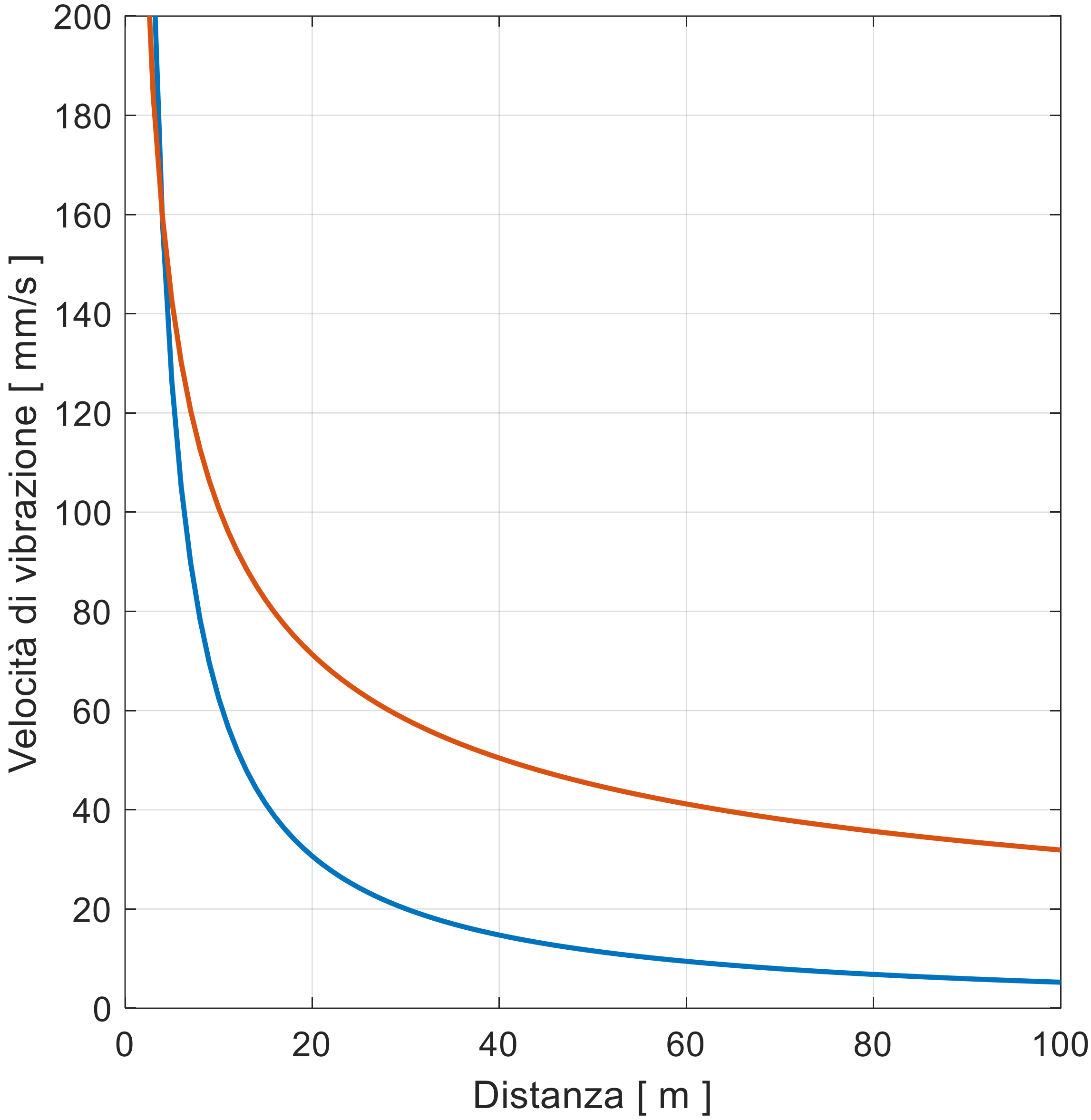


Figure 18 Estimation of the vibration velocity (peak particle velocity) versus the distance from the impact zone. The blue curve represents the shear waves and the orange one the surface waves.

Another way to appreciate the variation of the value of the vibration with the distance is the evaluation of many compaction tests or impact tests. The resulting diagram links an energy factor with the likelihood entity of the peak particle velocity. 100 meters far from the source zone, considering the category 4 to describe our soil, we may observe that the obtained value is comparable with those plotted in figure 18.

So, from figure 19 is possible to retrieve the value of 40mm\s for a distance of 100m:

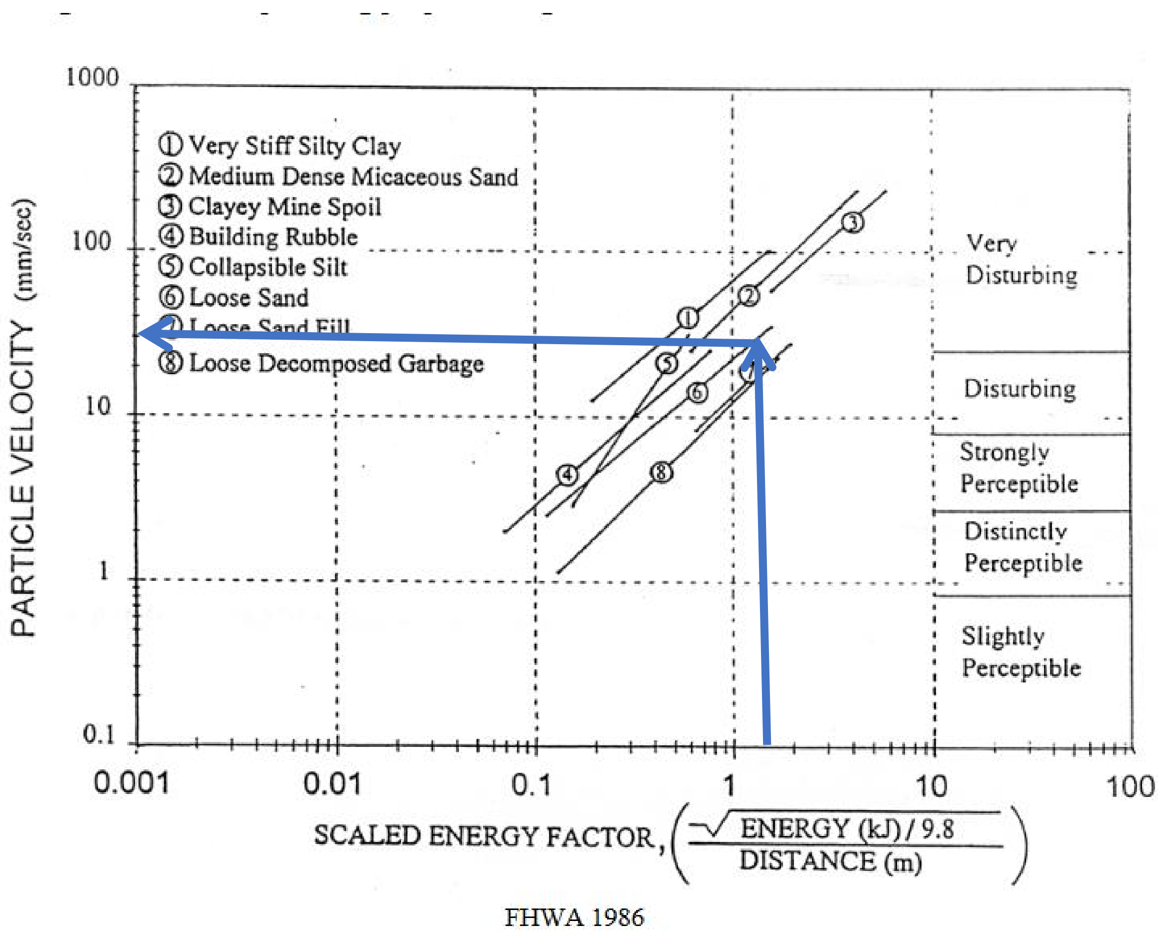


Figure 19Empirical plot which puts in correlation an energy factor with the peak vibration velocity, it is possible to see, following the arrows that the value obtained is similar to that obtained in figure 17

That is why is necessary to take an action to decrease the impact vibration velocities. There could be many actions, for examples:

- A pillow in very loose material (E<40MPa)
- A double layer of HDPE
- A pillow in combination with HDPE

We chose for simplicity the first one.The presence of a pillow of sand chances the physic of the problem. Now the first layer is not anymore, the upper layer of the ground. Now the layer where the deck is impacting is the sand. Firstly, we must define the characteristics of the pillow. From table 4 we can choose a material with properties which fit we our needs. The third material, loose material, presents a good ability in dissipating the impact energy. Its properties are here resumed:

Material	Coefficiente K	Tempo di impatto [s]	Deformazione [m]	Pressioni impatto [kPa]
Loosen material	25	0.20	1.2	300

As loosen material we choose sand, its properties are listed below:

- Young's modulus E = 40 MPa
- Shear modulus: G = 15 MPa
- Poisson's coefficient: ν = 0.3
- Density: ρ = 1500 kg/m^3

Now we have to consider the height of the pillow. We will try with 3 meters and 5 meters, that which fits better we our needs will be chosen. In any case, some general assumptions on the impact have to be done. After the impact of the deck with the ground we observe a permanent displacement of the ground surface, this it is seen in the figure 20. This depends from the elastic characteristic of the ground. Our consideration of a pillow with finite stiffness we can observe a deformed zone with curvilinear shape. The shape changes in functions of the material of the pillow. In the figure 14 is possible to see the different curves for different materials.

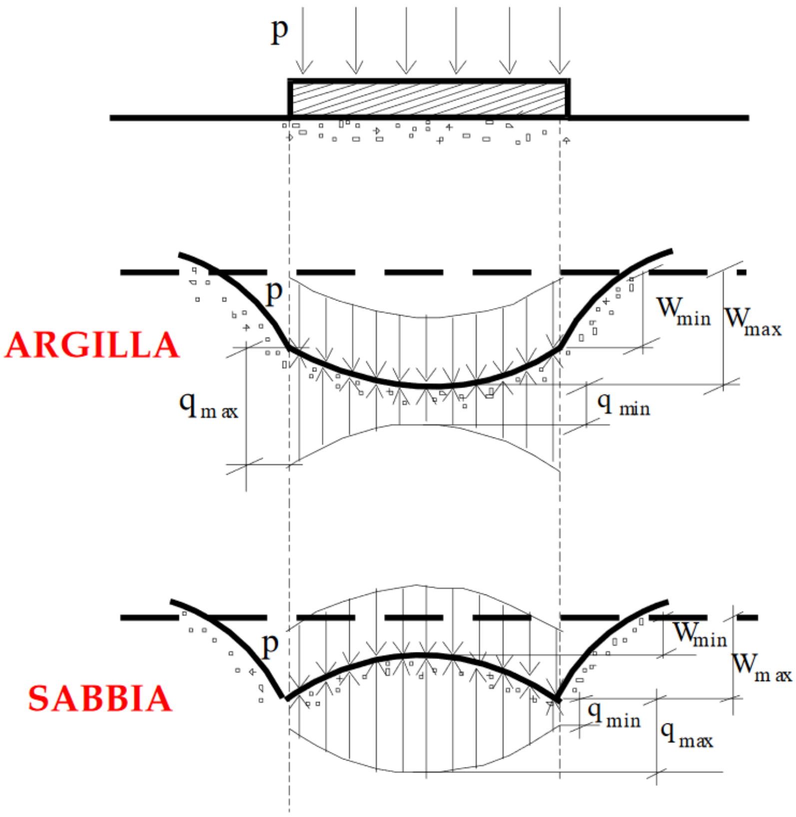


Figure 20Example of ground response with the hypothesis of a deck with infinite stiffness, from the upper, a ground with high stiffness, in the middle a clay, in the bottom figure is represented sand.

The stiffness of the pillow influences the shape of the deformation. The difference between clay and sand is that: pressures are constant with depth for clay with constant confinement while for sands which presents increasing stiffness with increasing confinement, impact pressures are higher in the middle of the pillow and lower at the border.

The generalized scheme is sketched in figure 21:

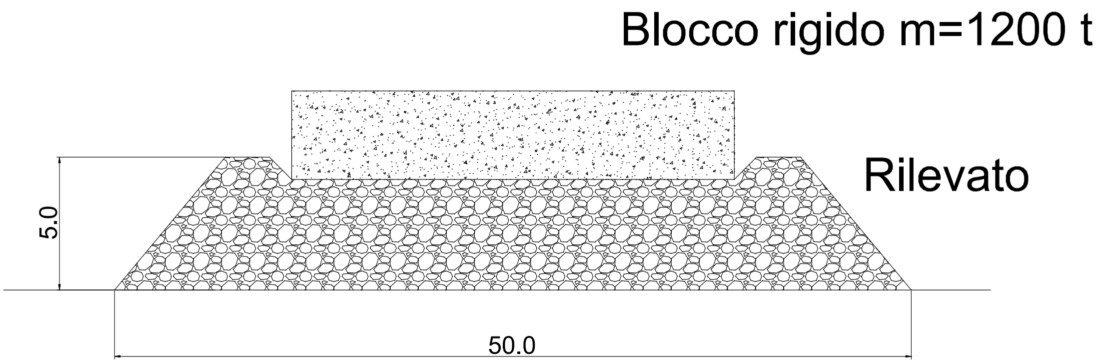


Figure 21 Represent the deck impacting on the sand pillow placed on the ground. In the picture are highlighted the height and the length of the pillow, they are, respectively 5 m and 50 m.

The equation to describe the model is the same as before:

$$m\ddot{z} + c_z\dot{z} + K_z z = F(t)$$

The Kz is changed, because are changed G and ν the new one is:

$$K_z = \frac{0.8 \cdot 42m \cdot 15 \cdot 10^6 Pa}{1 - 0.3} = 7.2 \cdot 10^8 \, {}^{N}\!/_{m}$$

Changing the stiffness will change also the dumping coefficient:

$$c_z = 0.2\sqrt{7.2 \, {}^{N}\!/_{m} \cdot 10^8 x 1200t} = 5.9 \cdot 10^6$$

The impact force is taken as before, F(t)=180000kN

The equation is:

$$1200\ddot{z} + 5.9 \cdot 10^6\dot{z} + 7.2 \cdot 10^8 z = 180000$$

The solution is analogous to that done before, and it is the sum of the homogeneous integral with the particular integral, we obtain:

$$z = e^{-2.4495t}(C_1 cos(24.37) + C_2 sen(24.37)) + 0.25$$

The boundary conditions remain the same:

$$\begin{cases} F(t) = 180000000N & t = 0 \\ v = 0m/s & t = 0 \\ z = 0m & t = 0 \end{cases}$$

Hence, the constant C1 and C2 are:

$$\begin{cases} C_1 = -0.25 \\ C_2 = -0.0251 \end{cases}$$

The general integral is:

$$z = e^{-2.4495t}(-0.25cos(24.37) + -0.0251sen(24.37)) + 0.25$$

The resulting plots are illustrated in figure 22 and 23.

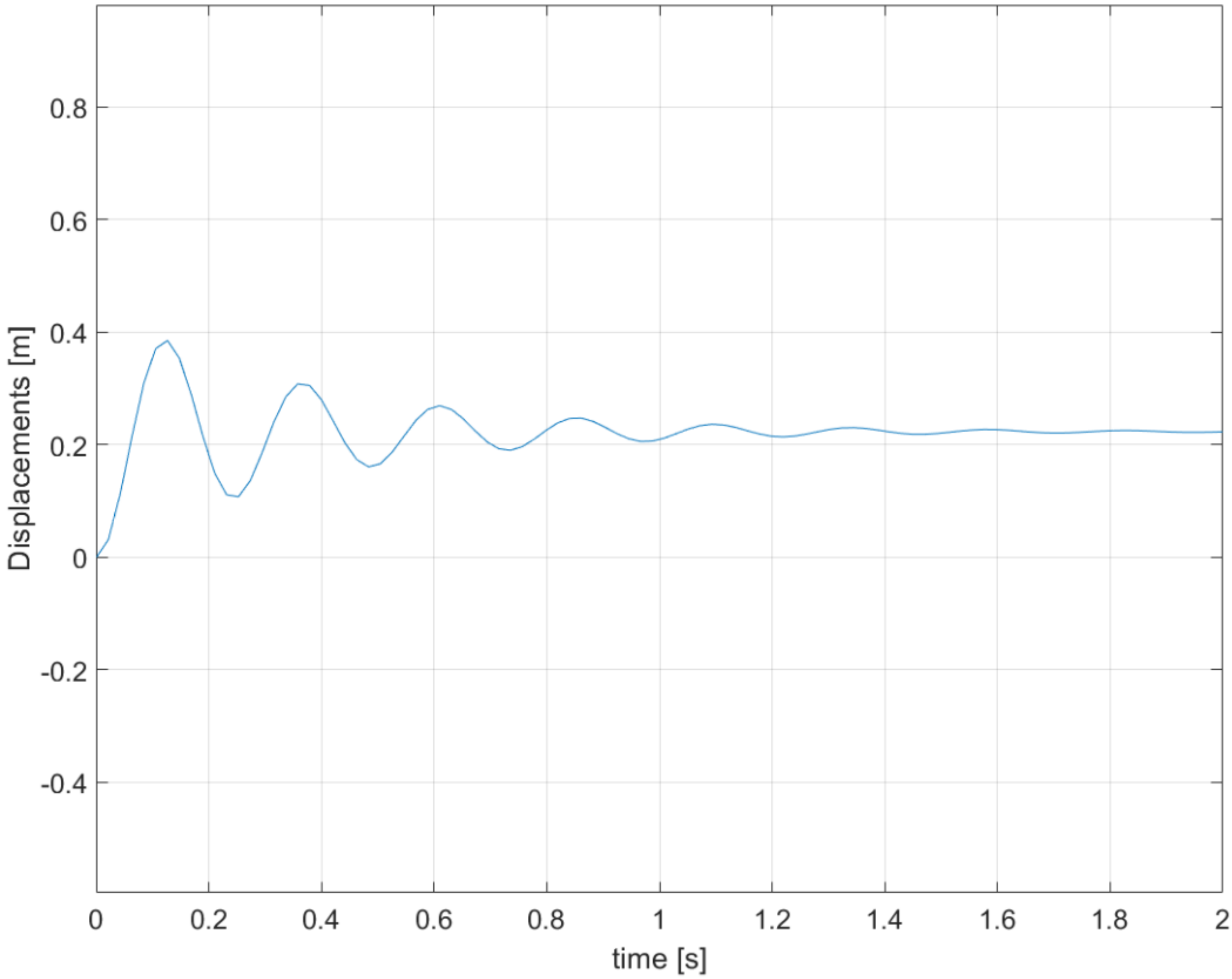


Figure 22 The plot represents the solution of our differential equation. In ordinates are reported the displacement in abscissae are reported the time. The maximum displacement is about 0.42 m in the early time.

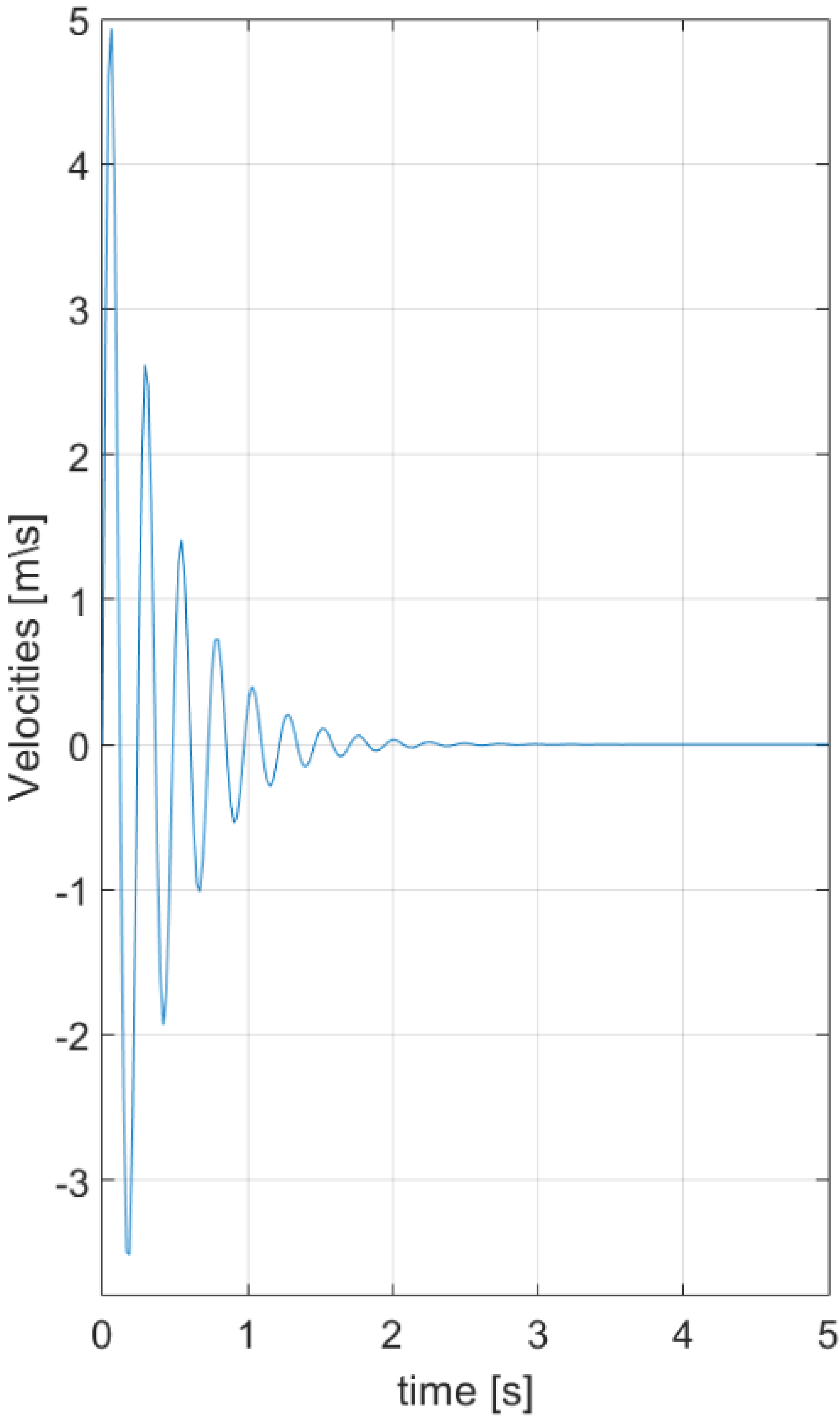


Figure 23This plot represents the first derivative of our solution. This time in ordinate we have the velocity. The highest value of the velocity in the plot is almost 5m/s

From the plot in figure 22, which represents the displacement due to the impact of the deck, we can see that, on sand, we have a higher vertical displacement. It reaches the value of almost 0.42m. Another increased value is the time before the oscillations stop. It is 1.5s almost. For the second graph, it in figure 23, it represents the vibrations at the impact. In this case the vibrations reach a value above 5m/s. They completely stop after 3 second.

A comparison between these two plots and those plots concerning the situation without the pillow could be useful to understand the situation and the variations. Firstly, we compare the displacement, secondly the vibrations plots. The comparison is done in figure 24 and 25.

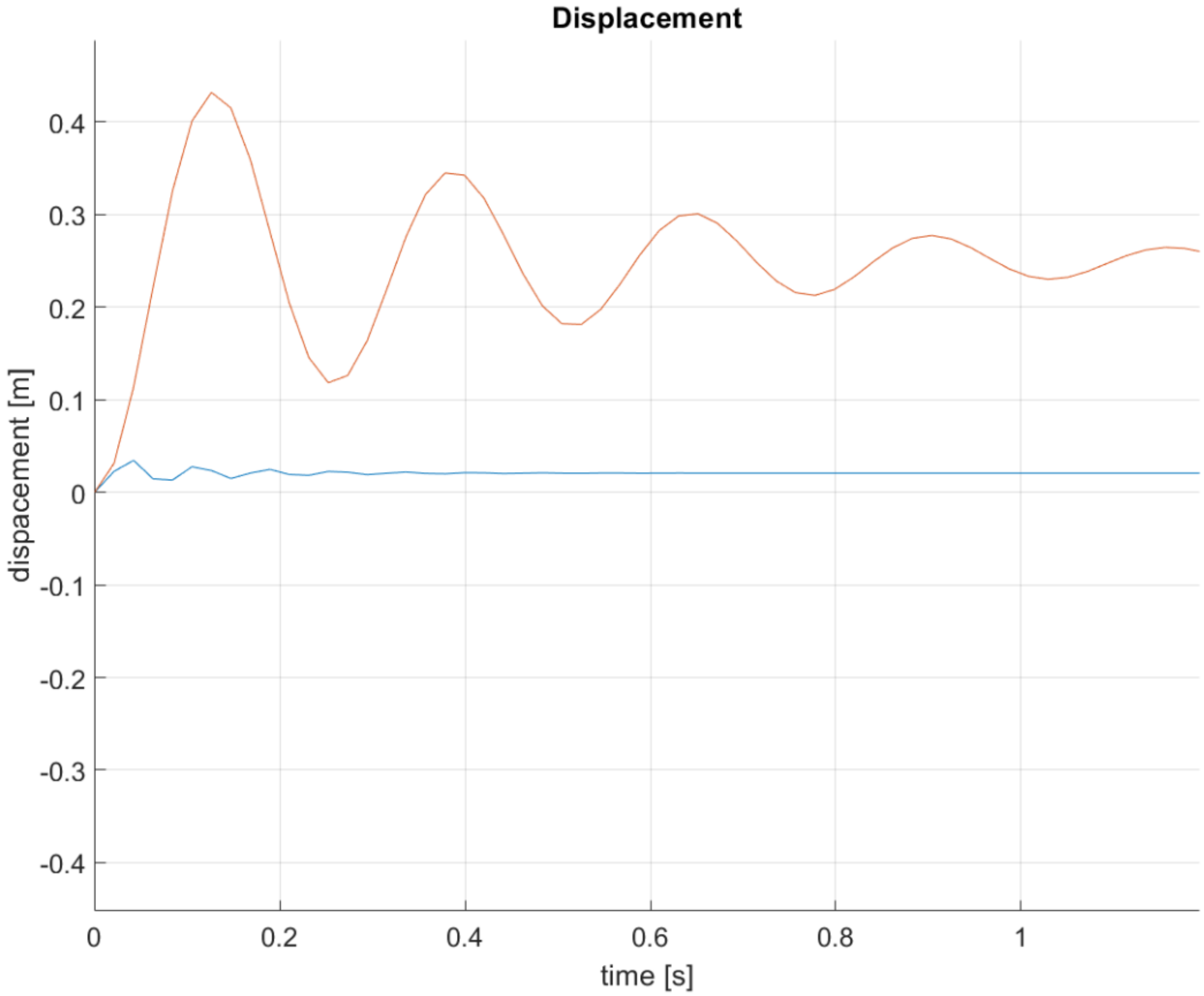


Figure 24Comparison between figure 12 and figure 16. The blue line represents di displacement shown in figure 12. The orange line represents the displacement in figure 16.

From the figure above is evidently how the stiffness and the dumping ration of a material influence the displacements. A thing which is not discussed before is the difference in the permanent displacement. For the ground it was around 0.025m. This is caused by its high stiffness. For the sands it tends to 0.25m. Almost 10 time greater. If we go to observe the stiffness of the ground and the stiffens of the sand, we can also appreciate that their values differs of an order of magnitude too. Also, the dumping coefficients differ about of an order of magnitude. This implies a higher oscillation time for the sand respect on the ground. This is confirmed by the picture above. The sand dumps the oscillation in more than 2 seconds, while the ground dumps the oscillations in just 0.2 seconds. It is possible to see that exist a linear proportionality between stiffness and permanent displacement. Also, it is possible to say that there is a linear proportionality between the time which the material needs to dump the oscillations and the dumping ratio. Let us look to the other comparison, that between vibrations velocities. It is shown in figure 25.

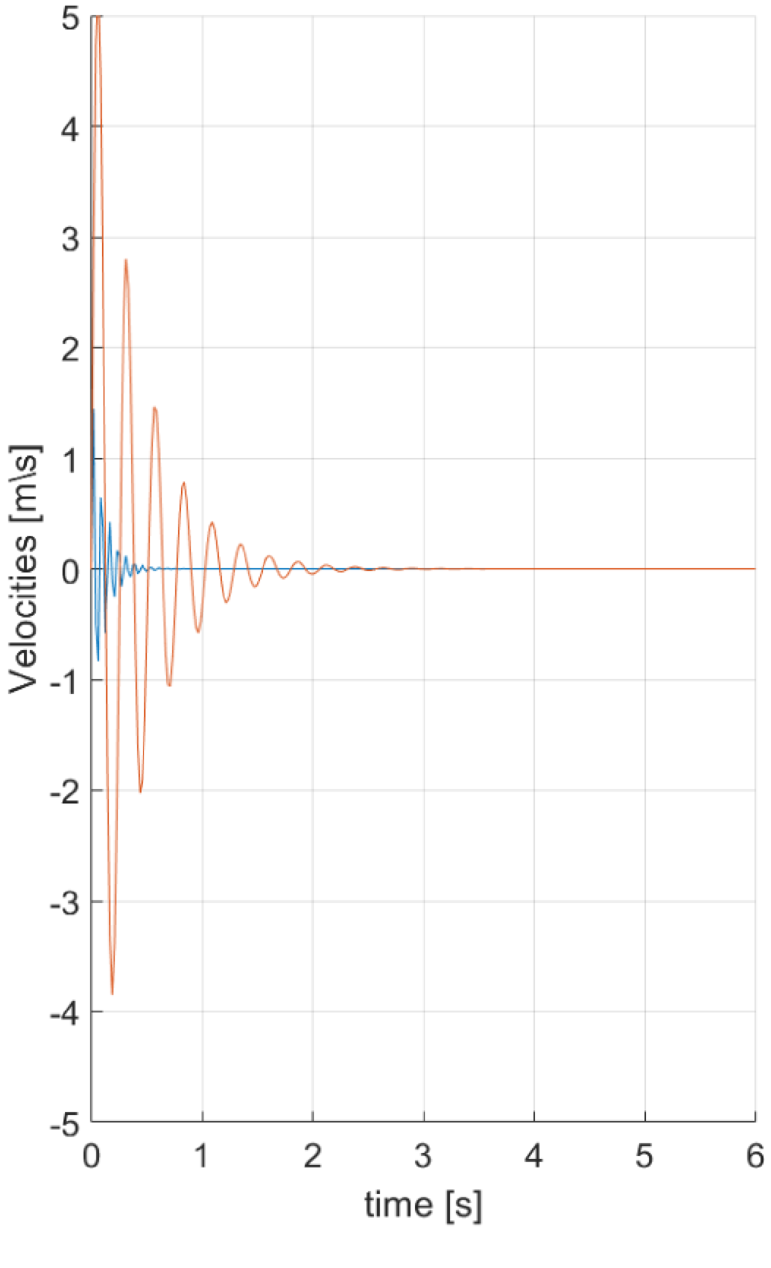


Figure 25 Comparison between figure 13 and figure 17. The blue line represents di vibrations shown in figure 13. The orange line represents the vibrations in figure 16.

It is possible to conclude that with the presence of the pillow of sand we obtain a huge dissipation of the impact energies. From figure 25 it is possible to say that the difference between the vibrations in sand and the vibration in the ground have values with a huge difference. Sands presents value bigger than ground, it means that more energy is dissipated from the sand and less is propagate to the strata below, in this case the ground. The difference with the case without pillow is that the ground receives a peak velocity of 5 m\s instead 30m\s relatives to the falling deck. Another confirm of this is the analysis of the behavior of the waves with the distance from the source. It is possible to see it in figure 26: From the picture is possible to see that from 30 meter of distance the value does not surpass the law limits.

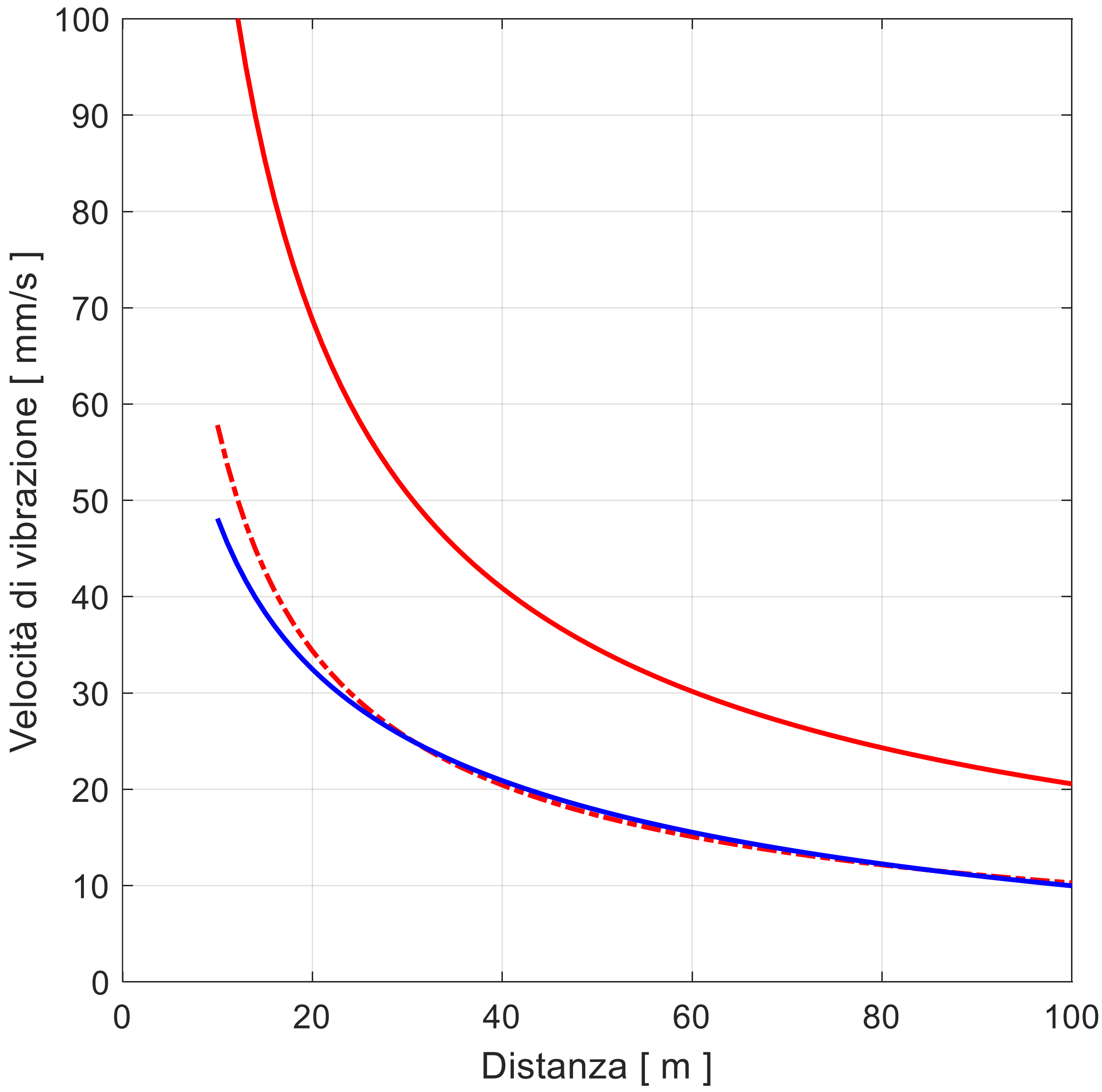


Figure 26 Simulation with the presence of the pillow. The blue curve represents the variation of the Rayleigh waves with the hypothesis of 1m of displacement. It is possible to say that the limit value, in this case is not overpassed. Vibrations are under 15mm\s for distance greater than 30m.

The last approach is trying to simulate the force as a sinusoid impulse. This is because it better represents the reality of the problem. The differential equation will be modified as follow:

$$m\ddot{z} + c_z\dot{z} + K_z z = F(t)$$

Where F(t)=180000kNsen(πωt).

ω represent the characteristically frequency of the oscillation, t is the time of impact.

The boundary conditions remain the same as in the previous case.

The solution of the equation has been modified as follow:

$$\sigma_1 sen(\sigma_4)\int_0^x A\sigma_2 cos(\sigma_3)F(y)dy - \sigma_1 cos(\sigma_4)\int_0^x A\sigma_2 sen(\sigma_3)F(y)dy$$

where:

$$\sigma_1 = e^{-Bx}$$

$$\sigma_2 = e^{By}$$

$$\sigma_3 = Cy$$

$$\sigma_4 = Cx$$

Where A, B and C are constants. The extremes of integration are the time domain, while the applied force varies just from 0 to 0,01seconds. The resolution of this mathematical problem goes over our goals.

The second part of this chapter consist in the roughly calculation of the stresses on the underground structures. In general, for an anisotropic distribution of the stresses, when the media is infinite linear elastic and the shape we are studying is circular. The physics of the system is explained in the figures below:

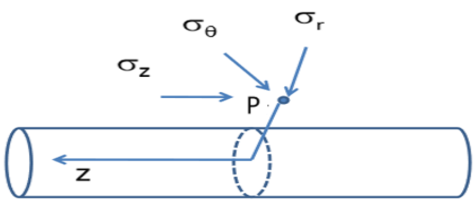


Figure 27This picture represents a horizontal cylindrical hole. We take it as an approximation of the pipeline going through the ground under the bridge. Z is the horizontal direction, σ_θ, σ_r, σ_z are respectively the tangential stress, the radial stress and the vertical stress acting of the point P

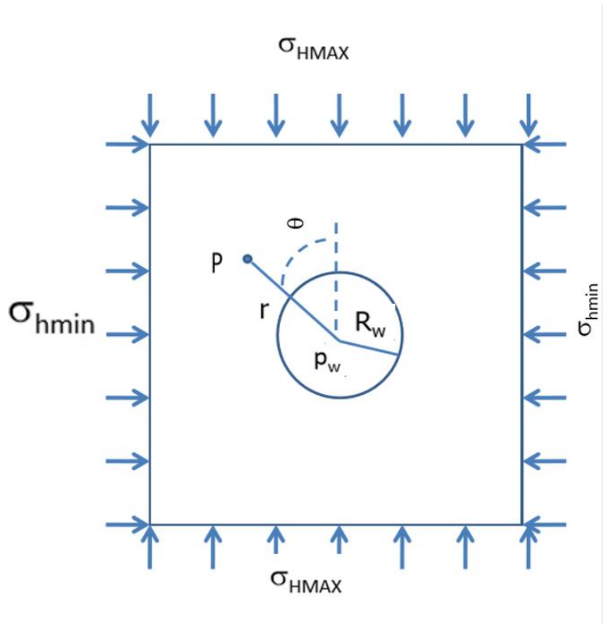


Figure 28 From this view is possible to understand the boundary condition applied to the rock surrounding the cavity. R_w is the radius of the void, r is the distance of P from the center of the void, and θ is the reference angle.

σ_{Hmax} and σ_{hmax} represents the anisotropic stresses, in particular the first represent the lithostatic pressure gradient and the second represent the horizontal stress, which is obtained by multiplying the vertical stresses for K_0. K_0 is taken equal to 0,5. σ_θ, σ_r, σ_z are respectively the tangential stress, the radial stress and the vertical stress acting of the point P. R_w is the radius of the void, r is the distance of P from the center of the void, and θ is the reference angle. In the following picture is possible to understand better how stresses act on a generic part of the rock.

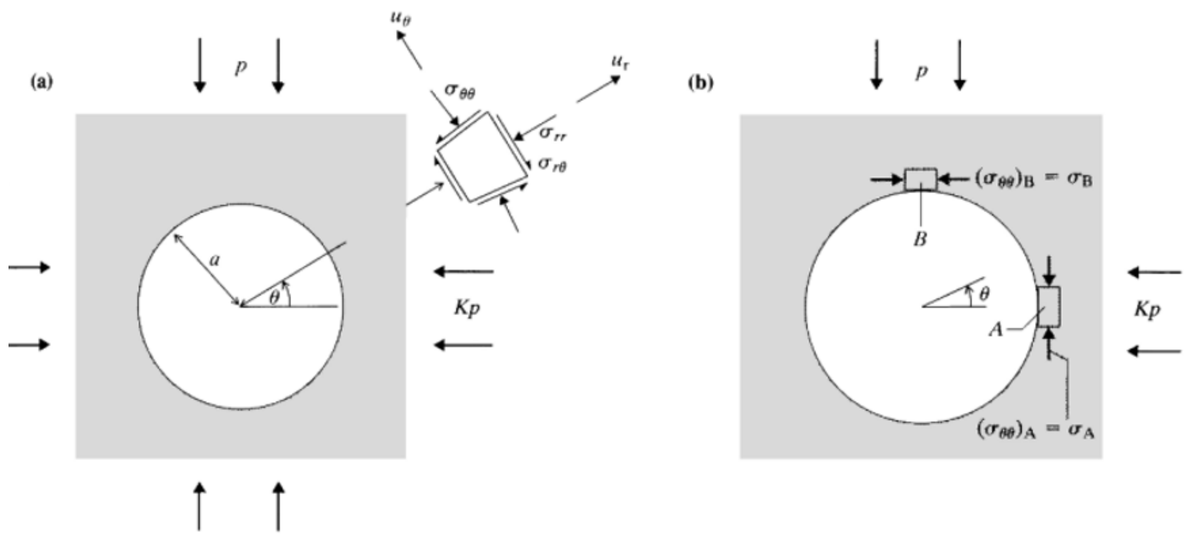


Figure 29 Here is well explained how the stresses act on a generic square of rock at a generic distance from the void. p=Lithostatic pressure, Kp=Horizontal stress, K=0,5 σθθ=tangential stress, σrθ=shear stress σrr=radial stress.

For the calculation of the tangential, radial and vertical stress we need to solve the Kirsh problem. The general solution of Kirsh problem is here exposed:

$$\sigma_r = \frac{1}{2}(\sigma_{h\min} + \sigma_{HMAX})\left(1 - \frac{R_w^2}{r^2}\right) + \frac{1}{2}(\sigma_{HMAX} - \sigma_{h\min})\left(1 + \frac{3R_w^4}{r^4} - \frac{4R_w^2}{r^2}\right)\cos 2\vartheta + p_w \frac{R_w^2}{r^2}$$

$$\sigma_\vartheta = \frac{1}{2}(\sigma_{h\min} + \sigma_{HMAX})\left(1 + \frac{R_w^2}{r^2}\right) - \frac{1}{2}(\sigma_{HMAX} - \sigma_{h\min})\left(1 + \frac{3R_w^4}{r^4}\right)\cos 2\vartheta - p_w \frac{R_w^2}{r^2}$$

$$\sigma_z = \sigma_v - 2\nu(\sigma_{HMAX} - \sigma_{h\min})\frac{R_w^2}{r^2}\cos 2\vartheta$$

$$\tau_{r\vartheta} = -\frac{1}{2}(\sigma_{HMAX} - \sigma_{h\min})\left(1 - \frac{3R_w^4}{r^4} + \frac{2R_w^2}{r^2}\right)\sin 2\vartheta$$

$$\tau_{\vartheta z} = \tau_{rz} = 0$$

Where pw is the hydrostatic pressure, if water is present.

Our interest is just on the surface of the pipeline, so when the r is equal to the radius of the hole, Rw=r. The solution of Kirsh problem takes this shape:

$$\sigma_r = p_w$$
$$\sigma_\vartheta = \sigma_{h\min} + \sigma_{HMAX} - 2(\sigma_{HMAX} - \sigma_{h\min})\cos 2\vartheta - p_w$$
$$\sigma_z = \sigma_v - 2\nu(\sigma_{HMAX} - \sigma_{h\min})\cos 2\vartheta$$
$$\tau_{\vartheta z} = \tau_{rz} = \tau_{r\vartheta} = 0$$

We consider pw=0, so we are working in dry conditions. The symmetry of the problem permits to evaluate the stresses just in two points. One on the top of the hole and the other on the horizontal section:

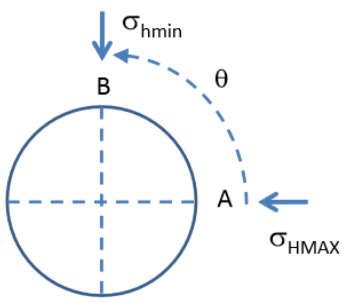


Figure 30 A and B are the point where the calculation of the stresses will be performed. The angle is 0 in point B, moves anticlockwise to A.

We analyze two cases:

Pressure due to the impact of the deck on the sand pillow:

$$F_1 = mg + \frac{mV_{imp}}{dt}$$

where Vimp=30m/s, m is 1200t and g is the gravity, dt is the impact time, we take it between 0,1 and 0,2s.

the equivalent impact pressure is obtained as follow:

$$P_{imp} = \frac{mg + \frac{V_{imp}}{dt}}{s} = \frac{m}{s}(g + \frac{V_{imp}}{dt})$$

The result of the calculation are:

- P(0.1s) =620000Pa
- P(0,2s) =320000Pa

At this pressure we add the pressure due to the sand pillow's weight. It is equal to 75000Pa. To obtain the equivalent pressure acting on the surface of the pipeline we must sum the three pressure components. The weight of the pillow, the impact pressure and the lithostatic pressure. To calculate the lithostatic pressure is enough to know that the density of the first layer is 1800kg/m^3, the gravity is 9,81km/s^2 and the depth is 2 meters. Hence p=1800kg/m^3x9,81km/s^2 x2 m=35316Pa. The results will be:

1. P equivalent (0.1s) =620000Pa+75000Pa+35316Pa=730316Pa
2. P equivalent (0.2s) =320000Pa+75000Pa+35316Pa=430316Pa

Now it is possible to calculate the stresses around the hole for both cases.

Casa 1)

Point A: θ=0, $(\sigma\theta\theta)_A=\sigma_A=P_{eq}(3-K)$

Pont B: θ=π/2, $(\sigma\theta\theta)_B=\sigma_B=P_{eq}(3K-1)$

$\sigma r=P_{eq}-2\nu(P_{eq}-KP_{eq})$

Tensioni tangenziali	[Pa]
σθθA	373987
σθθB	1869935
σr	467484

Case 2)

Point A: $\theta=0$, $(\sigma\theta\theta)_A=\sigma_A=P_{eq}(3-K)$

Pont B: $\theta=\pi/2$, $(\sigma\theta\theta)_B=\sigma_B=P_{eq}(3K-1)$

$\sigma r=P_{eq}-2\nu(P_{eq}-KP_{eq})$

Tensioni tangenziali	[Pa]
σθθA	223987
σθθB	1119935
σr	279983

Those stress values are in accordance with our prevision. We must say that those value are calculated considering the Kirsh problem and so have a meaning purely qualitative. In any case they give us an idea of the order of magnitude. To understand better the problem, we will move through a numerical resolution of the problem.

3.2)Numerical model

The second step in the calculation of vibrations and displacement is the resolution of a numerical model. At this point we want to prove the results obtained with the analytical procedure. In general, the numerical model has to give us the same or comparable results of the analytical one. To simulate our model, we must define a geometry of the problem. The geometry we take in consideration is that described in table 2. It means the geometry of the ground where the deck will impact. From table 2 we may see there are three layers. Each one of them is thick 10 meters. The other two geometrical parameter, the width and the length, are given by us. So, we suppose three cuboids, stacked one over each other. They function is to approximate the first three layers of the ground under the pillar. The length and the width taken in consideration are respectively 200m and 15m. Those two measures come from a qualitative evaluation of the geometry for a good point of application of the impact pressure. The geometry that comes from is a kind of 1D model, ant the result is shown in figure 20, here below:

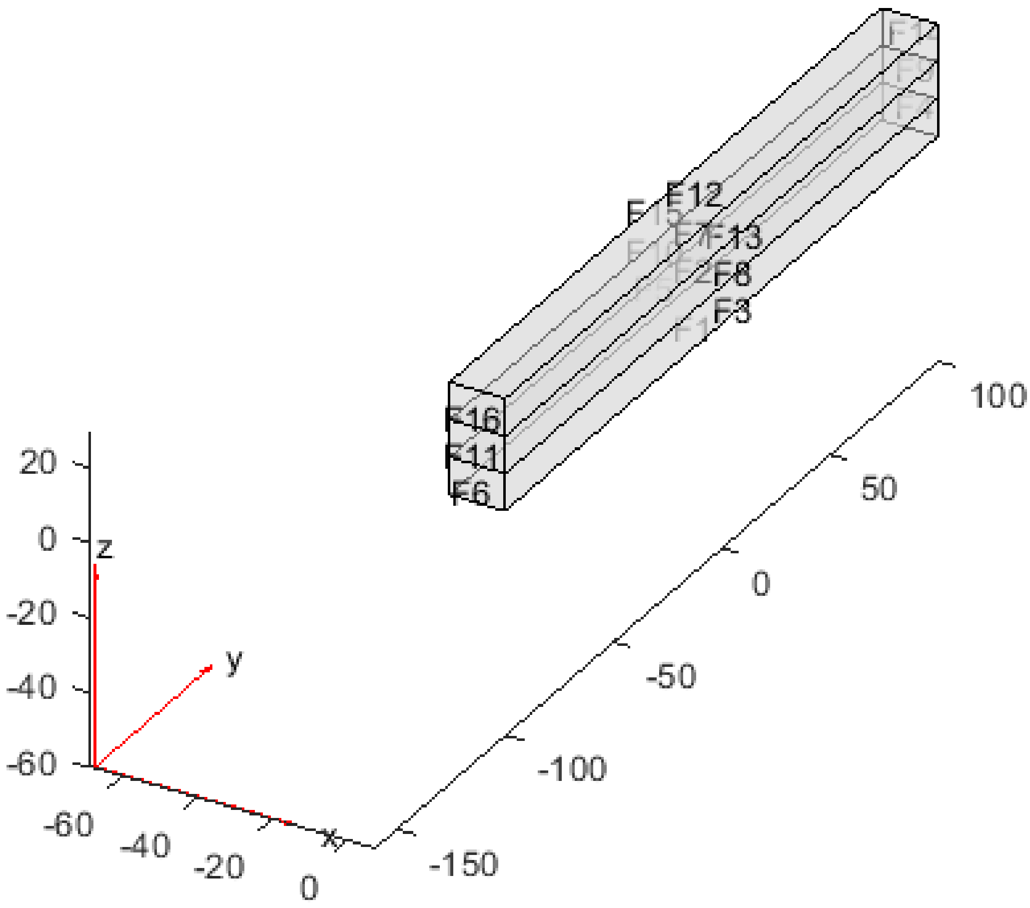


Figure 31In this picture are represented the three layers of the ground one over each other. Each one of them has the same thick, equal to 10m. The total depth reached is 30m. Their length, along the y axes is 200m. The width, along the x axes is 15m. The faces represent the possible location where we may apply the pressures.

Once produced the correct geometry we need to give the right properties to each stratum. Will be necessary the categorization of the ground done in chapter Geophysical parametrization.Table 2 and Table 3 give us the elastic parameters of the ground. To do a good approximation of our model is needed also the parameter concerning viscoelasticity, they will be treated later. Firstly, we develop a model completely elastic. In this case we do not need any viscoelasticity parameter. To develop this model, we use Matlab's Structural Mechanics tool. Then we need to give to each layer the right properties. The three parameters we need are E and the ν are listed in tab 3 while the density is reported in table 2. Once given the correct properties is possible to give the boundary condition. We suppose the multicuboid with a fixed constrain at face 1 that is the lowest face. All other faces are supposed to be free. Our idea is to let all the other surfaces free for shear, displacements and strains.

In this way we supposed to have a good approximation of the phenomena. The face 1 is supposed fixed to the bedrock.
Now, we must simulate also the force acting on the surface. We have to choose the face where we want to apply the forces. In this simulation is not possible to apply forces on to the edge, but just on the faces. The perfect choice would be the application of the force on the edge between face 12 and face 16. So, we supposed to have the impact over the face 16, hence, outside our domain. In this case, the propagation of the pressure will arrive, in our model, all from face 16. In practice we simulate, for this model, to have the pressure applied just at the face 16. The correct representation of the applied force or in this case of the applied pressure should be an impulse. Considering the elastic nature of the ground this is impossible, so we should choose a gaussian distribution. Hence, we should model the applied pressure like a gaussian as follows in figure 32.

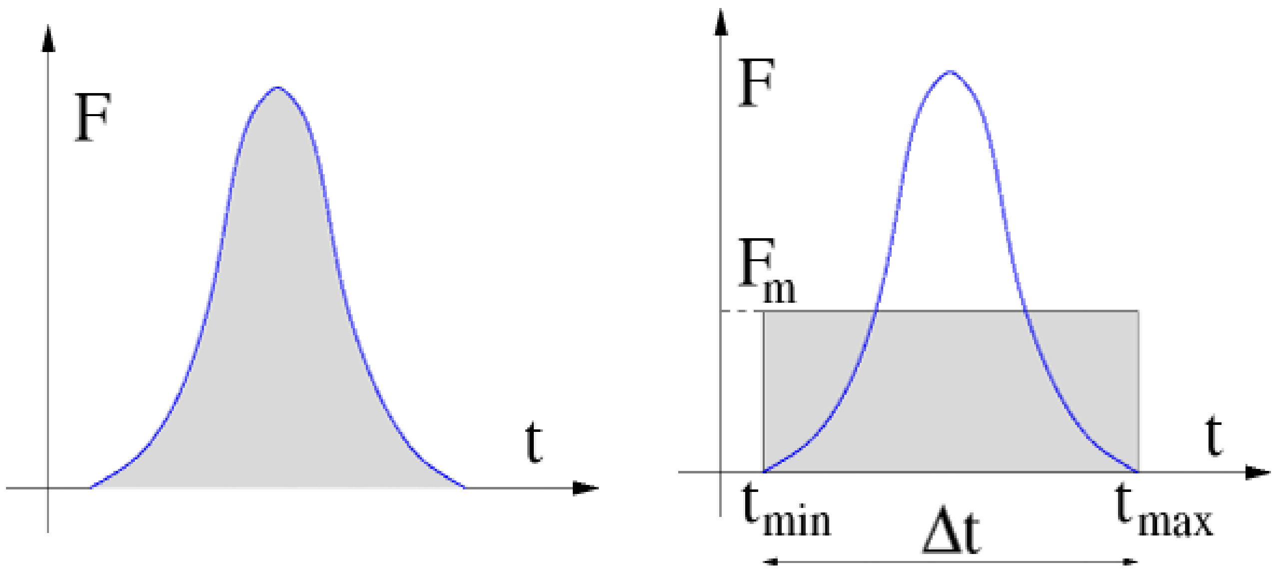


Figure 32 This picture represents the gaussian distribution of a generic force. The plot on the left shows the area underlined by the force function. In the plot on the right we have another possible representation. The area in the rectangle is the same to that in the figure on the left, so we can represent the gaussian distribution with a average function.

For this tool of matlab it is impossible, to model an applied force like a gaussian. Hence, we modeled the pressure as shown in the plot on the right in figure 32, thus, a rectangular function. The consideration done is that the area underlined by the gaussian is the same to that of the rectangle drawn considering an average value of pressure-force. The effect of the maximum value of the force, represented by the pick of the gaussian, will have a worse effect on the ground than an average value of the force. In any cases it is impossible to represent it as a gaussian. We consider the application time equal to 0,01s. This time is in accordance with the empirical results obtained with the tumping.

So, to define the impact pressure we use the start time and the end time, as shown in figure 33:

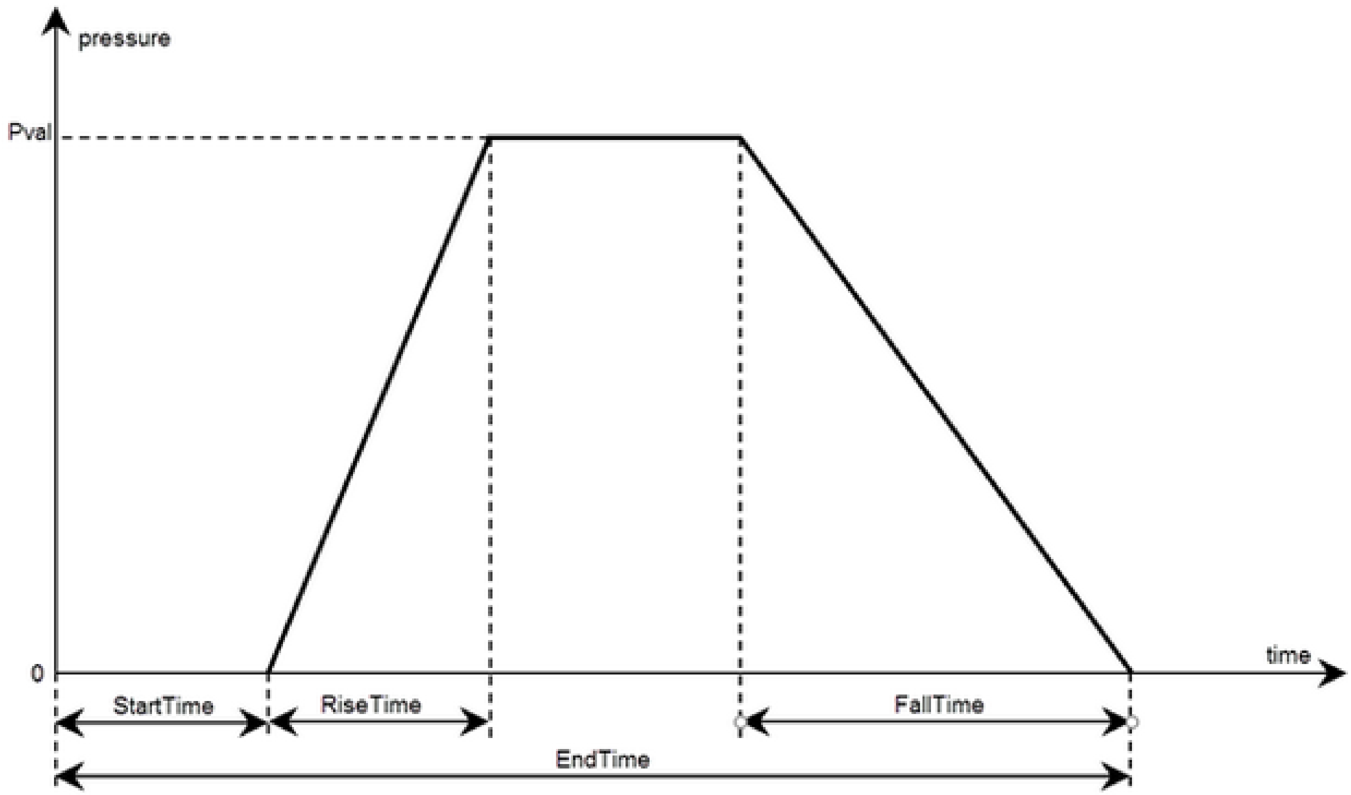


Figure 33 In this graph is represented a general characterization of a pressure. The start time represents the time when the application of the pressure starts. The risetime is the interval into which the pressure reaches the maximum value, the fall time is when the pressure goes from the maximum value to 0. The end time represent the finish of the application.

For simplicity we did not consider the pressure function as a trapezoid, but easily as a rectangle, like in figure 32. So, we gave the start time equal to 0 seconds. Then we gave directly the end time equal to 0.01 seconds.

Once imposed all the physical parameters it is possible to generate the mesh. The mesh is needed to guarantee the accuracy in the representation of the physical phenomena. Hence, to define it density we must understand the needed vertical and horizontal resolution.

We are analyzing seismic waves, pressure, Rayleigh and shear waves. From table 2 we have, for each of the strata its pressure velocity and shear velocity. The Rayleigh waves are not reported, we may suppose them in between of the previous two categories.

The stratum with the lowest velocities is the upper. Its velocities are:

1. V_p=800m\s
2. V_s=300m\s

For Rayleigh waves we may suppose 350m\s.

The dominant frequency of the system, f, is about 10Hz. For the calculation of the vertical resolution needed we must calculate the wavelength of the wave.

$$\lambda = \frac{Vs}{f} = \frac{300m\backslash s}{10Hz} = 30m$$

The, we can take the wavelength and divide it for four:

$$\frac{\lambda}{4} = 7.5m$$

The resolution should be at least 7,5 meters. The general mesh furnished by the matlab tool does not cover this length, is too big. So, we need to dense it.

In general, the resolution we calculate for the mesh could be thinner and thinner. To approximate better the shape of the sinusoid is better at least to subdivide the wave in 15 bins.

In this way we obtain the net of our mesh really thin and so a good approximation of the event. At the end we choose a mesh with the bins wide 50 cm each.

Our model is long 100 meters, that means will be present, in our mesh, 200 elements in length. In depth, instead, the geometry is deep 30 meters, so, 60 squares. And, for what concerns the width we have just 15 meters so 30 bins.

The desired result is shown in figure 34, here below:

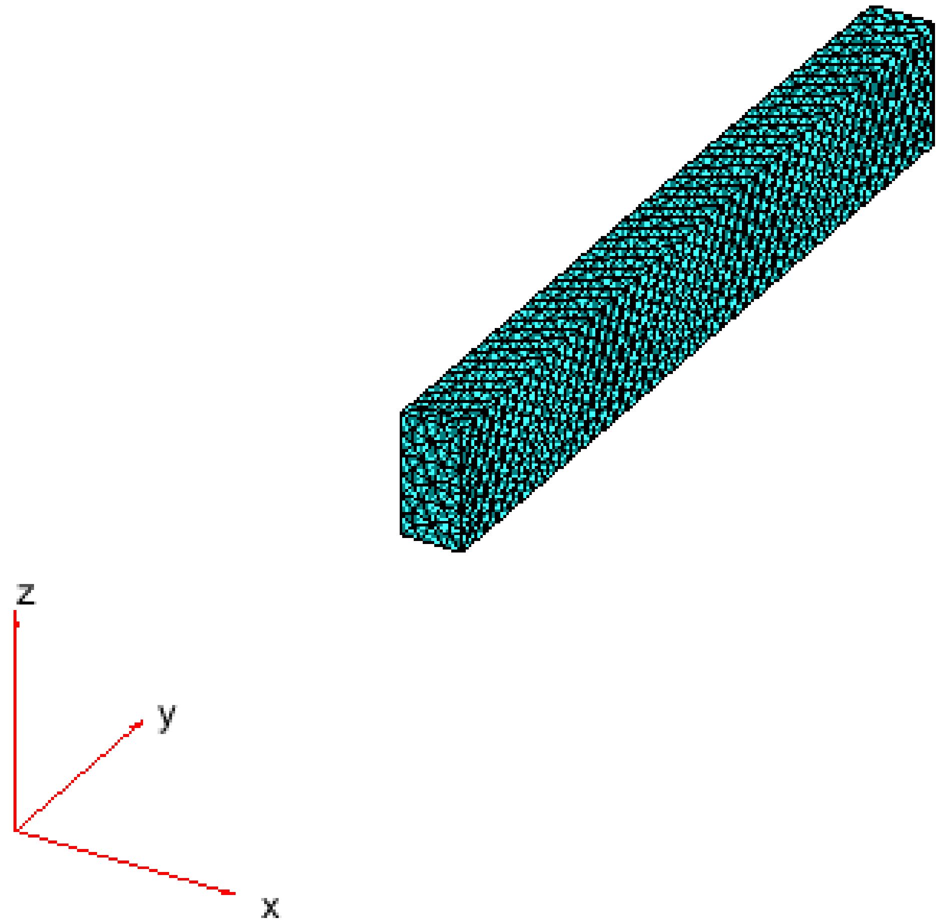


Figure 34 This figure represents the mesh of 0.5m covering the multicuboid represented in figure 20

Another important part of the program is the input of the initial conditions. This means the conditions before the impact, at the instant 0 seconds. Those are necessary to solve the problem has seen in the analytical part. That is why the initial condition will be the same chosen for the analytical model, so:

$$\begin{cases} F(t) = 180000kN & t = 0 \\ v = 0m/s & t = 0 \\ z = 0m & t = 0 \end{cases}$$

At the instant 0 second we have that the velocity and the displacement at the interfaces are equal to 0, also we have the application of a constant force on the face 16.

The last part is defined the time interval on which we want to observe the phenomena. It is also important to choose the number of frames into which we want to subdivide that time. Considering that the application time is 0.01, we take a line space of 0,15 seconds to see all the propagation moving. We divided it in 5 frames.

Hence, we have:

0 0.0375 0.0750 0.1125 0.1500

The instant 0 seconds represents the initial conditions of the systems, it is represented in figure 35:

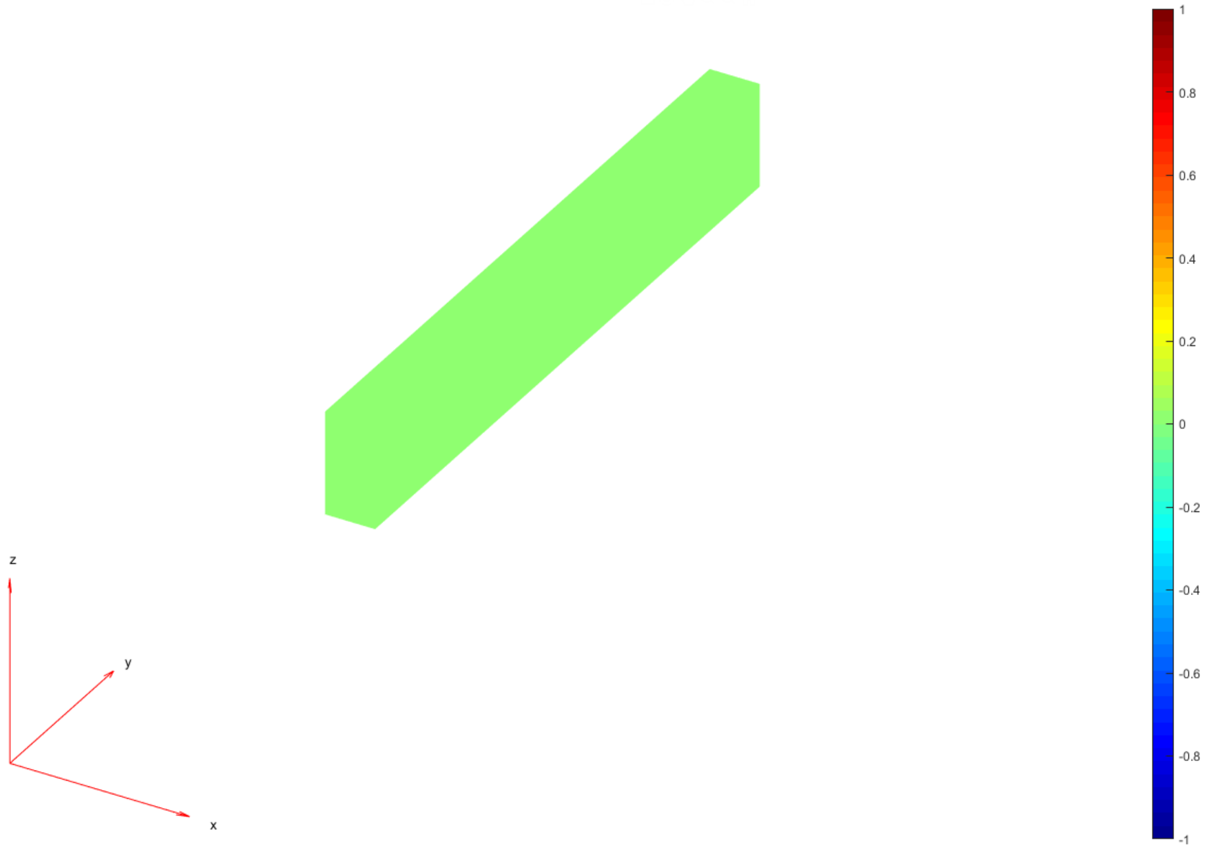


Figure 35 Displacement at time 0, the scale at the right size represent the displacement in m. Everything is green, thus, we have 0m of displacement in all the structure.

In the figure above everything is green that represent the initial condition equal to 0. Hence, the interesting frame is that between 0s and 0.0375s. This frame is important because represent the impact of the deck on the ground. The result of the second frame is in figure 36.

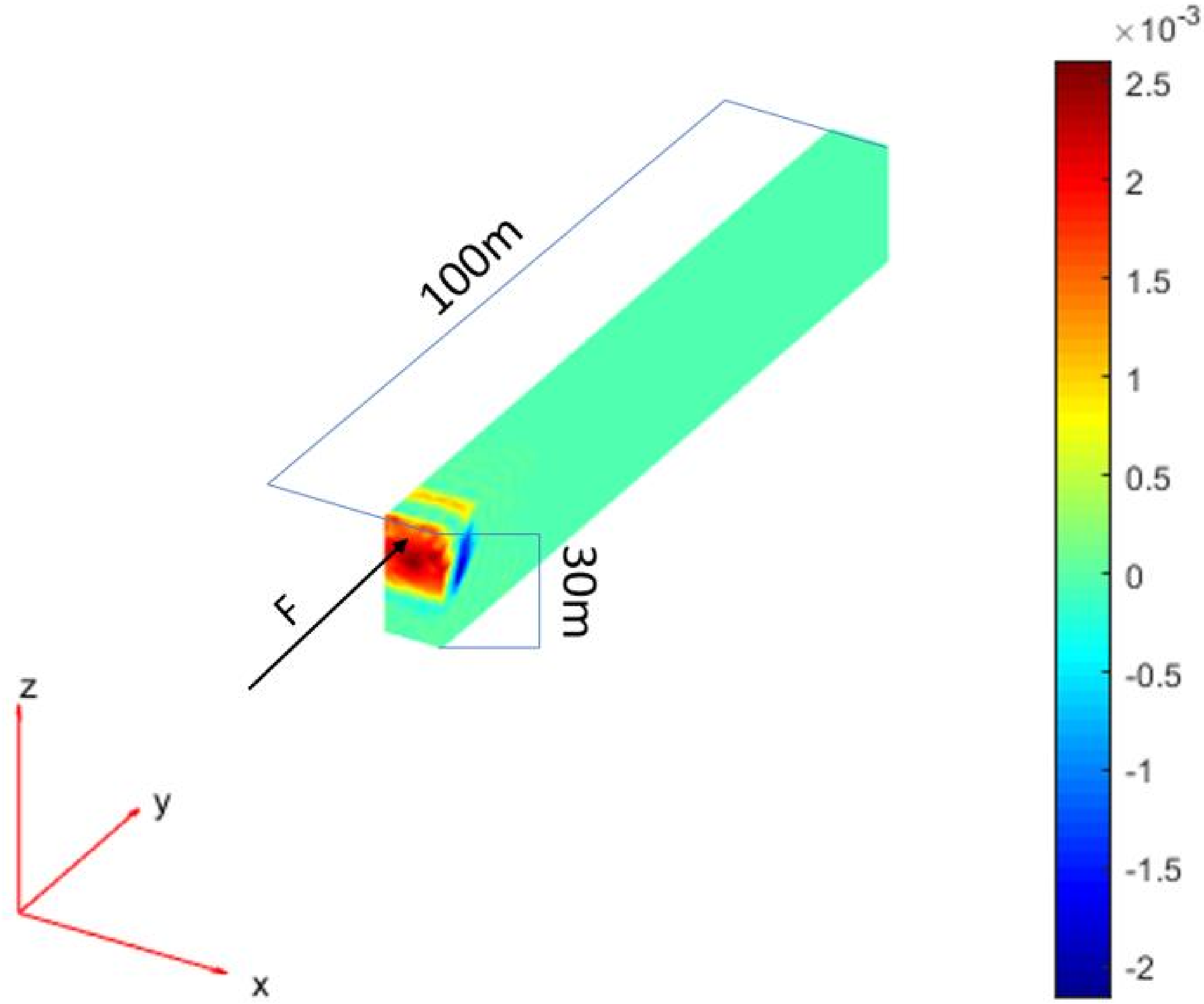


Figure 36 The pressure is applied on the face 16. The scale on the right is in m. This frame is the frame at 0.0375s.

From the second frame it is possible to see the impact of the pressure. In the near impact area, we can notice the highest values of the displacements. They are, at their maximum value, more than 2,5 mm. The red area represents the compressed area. There is also a zone subjected to traction. The compressed zone arrives until the first 10 meters of the first layer. The second layer is reached by the deformation. Just the very near surface of the second layer is reached. From table 2 and table 3 we know the properties of the two layers. We can say that those two layers differs just for the velocity of the pressure waves. In the second layer the velocity is fast. Looking at the plot is evident how the propagation goes faster in the second layer. To better appreciate this phenomenon is better to change the scale and to plot in detailed the geometry. It is possible to see it in detailed in picture 37. The figure represents di y-z cross section. From here it is possible to see how the propagation goes. We might understand the propagation of the phenomena looking at the velocity's values in table2. The second frame is at 0.0375s. With our data it is possible to estimate where the wave front is after this time. Then, compare the estimation with the plot. The estimation is collected in the following table:

Table7: Estimation of the wave front path

layer	Vp [m/s]	Vs [m/s]	Y of Vp [m]	Y of Vs [m]
1	800	300	30	11.25
2	2000	300	75	11.25
3	2500	400	93.75	15

From the table it is possible to say that pressure waves might reach the entire depth of the geometry, which is 30m. 30 meters are reached in each direction because the wave front is isotropic. Shear waves, which are slowly than P waves, might reach just the interface between the first and the second layer. To understand if those prevision are good we can look at picture 37.

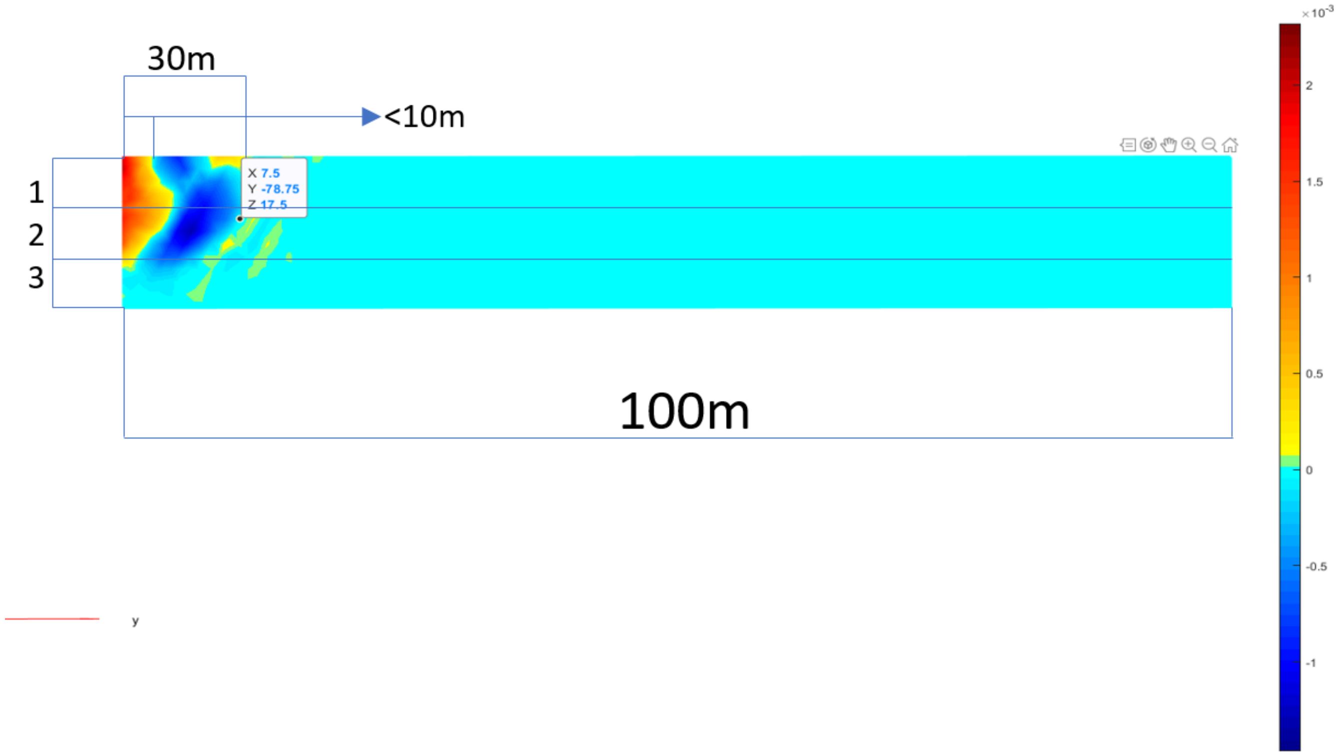


Figure 37Particular of the picture number 16. The scale is in m. The axes start from the lower wedge. To calculate distances, depth or width is necessary to do the subtraction, 100 minus the read quantity

As it is possible to see in picture 17 the displacement, for first layer, arrives in 0.0375s at almost 30m from the origin of the applied stress. It is represented in blue. This means we are looking at pressure waves front, in accordance with the prevision. The red one, which reaches, for first layer, almost 10 meters, could represent the shear waves front, in accordance with the calculated value. In the second layer we have the blue front at almost 30meters. If we make some consideration regarding the time it is possible to understand that the pressure waves arrive in the second layer after 0.0125 seconds. When they arrive at the interface between the first and the second layer the velocity of the pressure waves increases. From this point they might reach the third layer in just other 0,005 seconds. Obviously, the path of the wave front is not spherical, this because the pressure is applied over the whole face 16. This is why the previsions done for the layers below the first do not fit exactly with the model shown in figure 37. The picture shows also the compressive zone and the tensile zone. Changing the color of the scale it is possible to appreciate the total propagation zone. In the figure below, the 38 is represented the frame at 0.075s. Here the displacement is arrived until 40 meters. The zone below 40 meters is subjected a negative displacement, which means a tensile zone. The displacements value is almost 1mm. In the vicinity of the application zone 2mm of displacement are reached. The Propagation is also arrived at all the second layer and reached the third. The figure represents di y-z cross section. From here it is possible to see how the propagation goes. We might understand the propagation of the phenomena looking at the velocity's values in table2. The second frame is at 0.0375s. With our data it is possible to estimate where the wave front is after this time. For

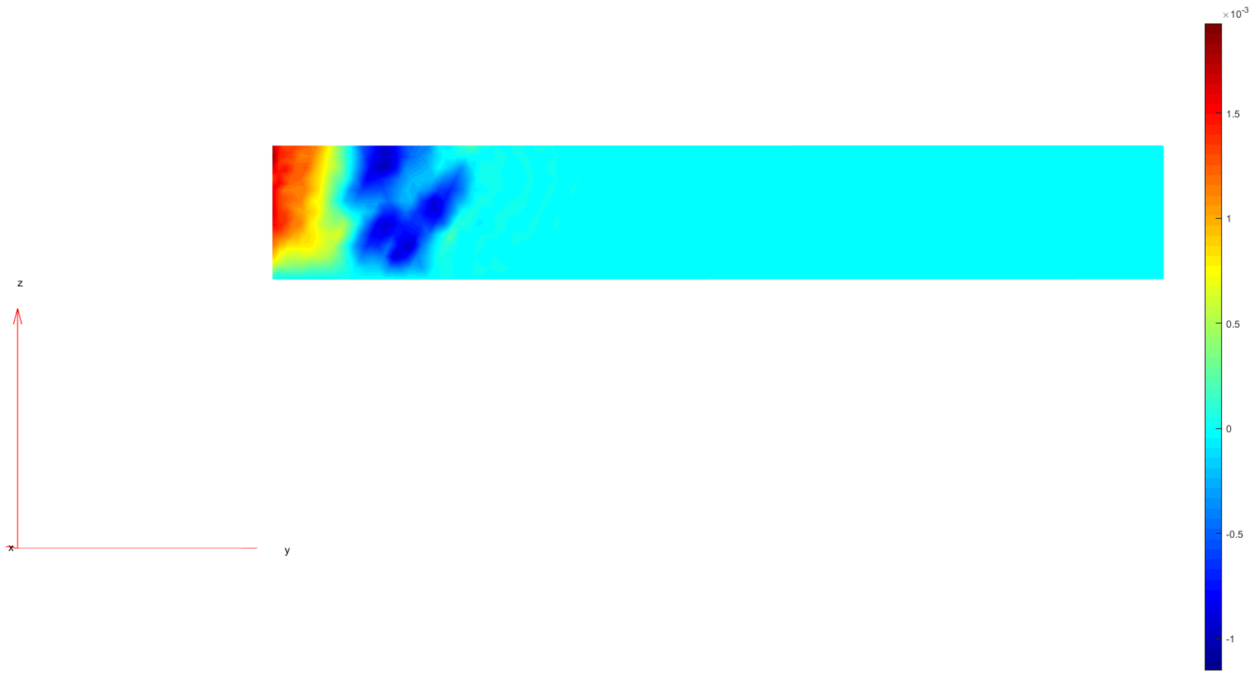


Figure 38 The pressure is applied on the face 16. The scale on the right is in m. This frame is the frame at 0.075s.

The third frame represented in figure 39 shows the instant at 0.1125 second. It is possible to see that the P wave front reaches almost 40 meters. The discrepancies with the prevision are wide. The S waves, which are represented by the red-yellow color arrives at almost 20 meters.

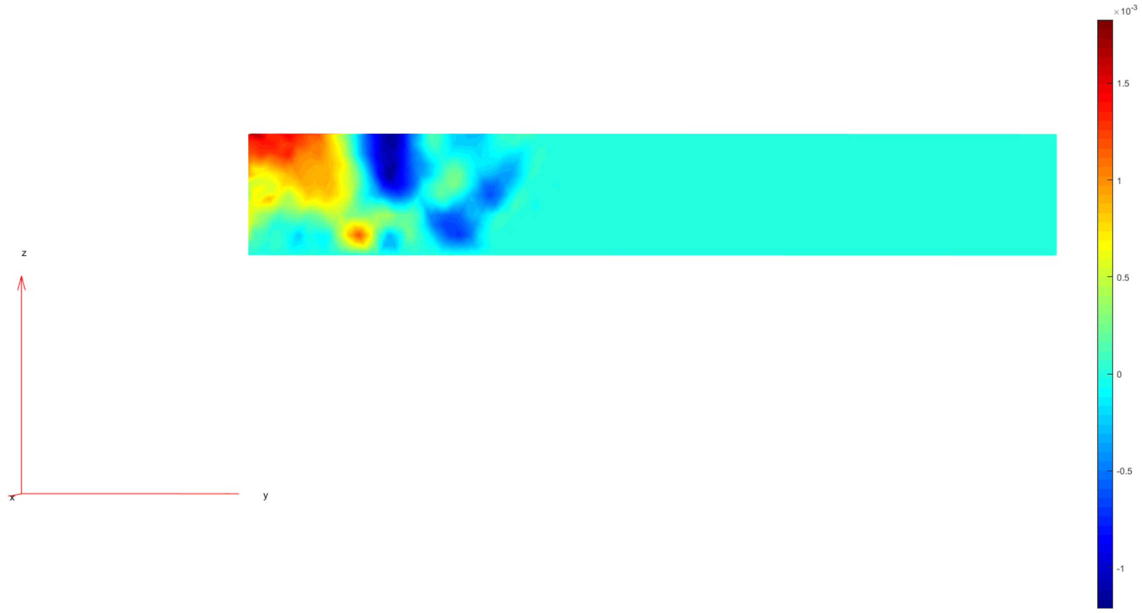


Figure 39 The pressure is applied on the face 16. The scale on the right is in m. This frame is the frame at 0.1125s

In the last frame we have the situation shown in figure 40. The application of the pressure is ended from 0.14 seconds. The wave front is completely detached from the face 16. Almost all the phenomenon is disappeared.

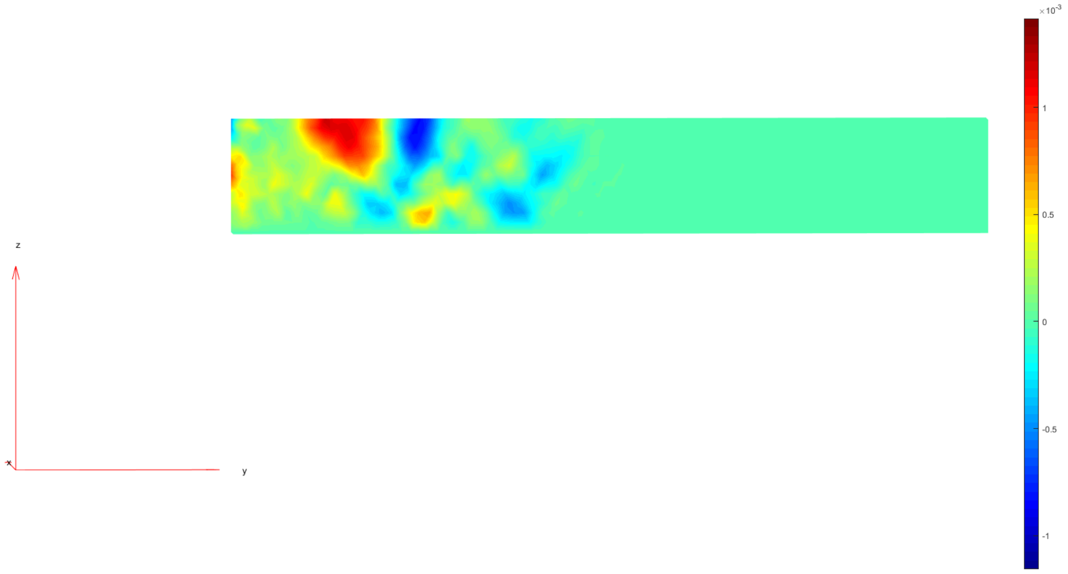


Figure 40 The pressure is applied on the face 16. The scale on the right is in m. This frame is the frame at 0.15s.

It is also possible to plot the velocity in place of the displacement. The first frame is represented in figure 41. It is possible to appreciate the P wave front. The blue one, in accordance with the hypothesis done until now. The phenomenon seen in the part where the pressure is applied could be made by a insufficient resolution for the time.

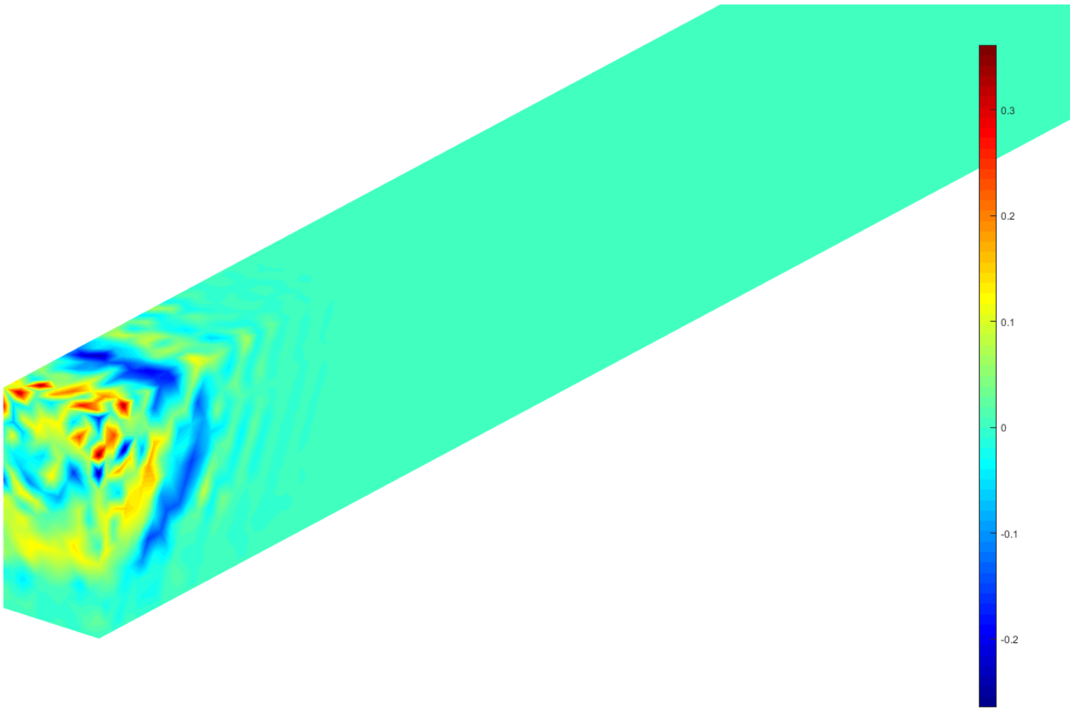


Figure 41 The pressure is applied on the face 16. The scale on the right is in m/s. This frame is the frame at 0.0375s.

From the second frame (figure 42) the spotted phenomenon increases.

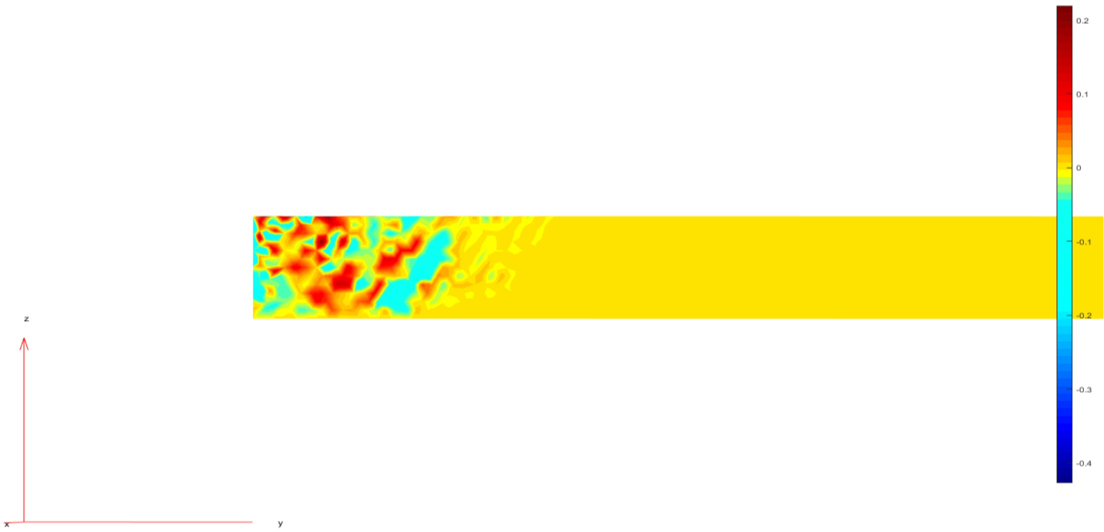


Figure 42 The pressure is applied on the face 16. The scale on the right is in m. This frame is the frame at 0.075s.

For the first model we applied the initial pressure the face 16. This approximation does not give us the correct information regarding the propagation of the velocities or of the displacements. It gives us just an idea. We need to apply the pressure on the surface of the ground. To do it we must pass through a model that is almost 1D. 1D is intended that two of the three dimensions of the cuboid are negligible in comparison with the depth.

We image to have only a layer, the first. This layer is supposed to be 100m depth. In this case impact surface is the upper part of the cuboid and is considered as bigger as the surface of the deck which is impacting. All the other considerations, properties, boundary conditions, initial conditions, are the same of the previous mode.

The geometry obtained is shown in figure 43:

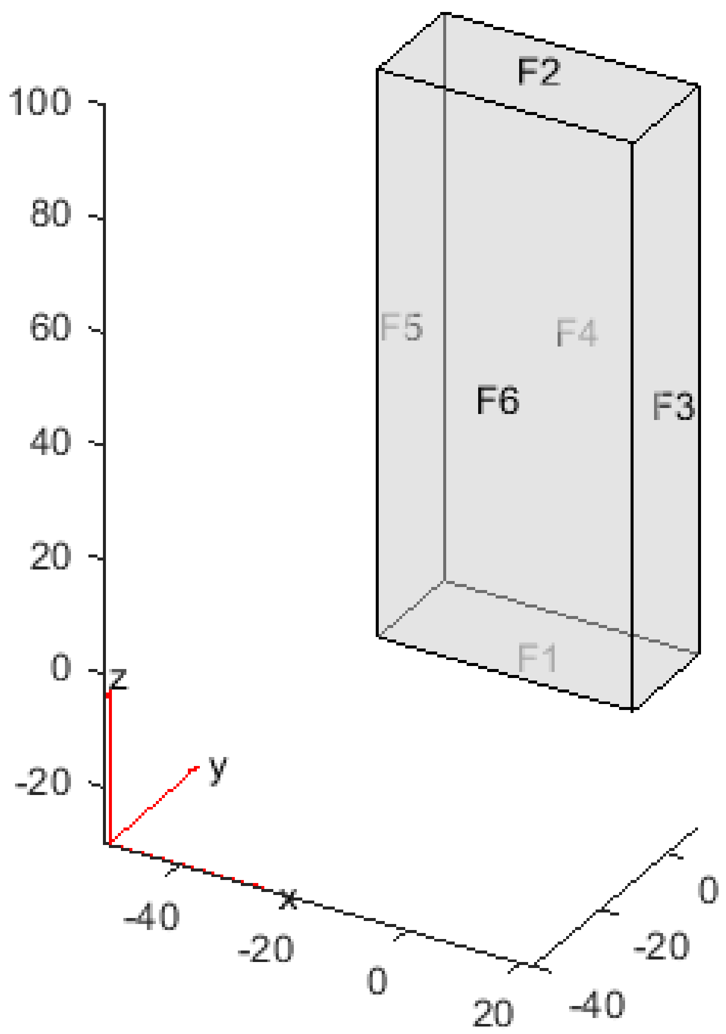


Figure 43 Geometry representig the firt layes wtth an immaginary dept of 100m. Its width is on the y axes and is equal to 15 meters, the length is on the x axes, equal to 42 meters. Those two length are those of the deck impacting of the face 2.

Here the application face is the F2.

Now is possible to cover it with the mesh. The mesh used for this model is the same to that used for the previous model. The consideration done for the mesh are the same done in the previous simulation.

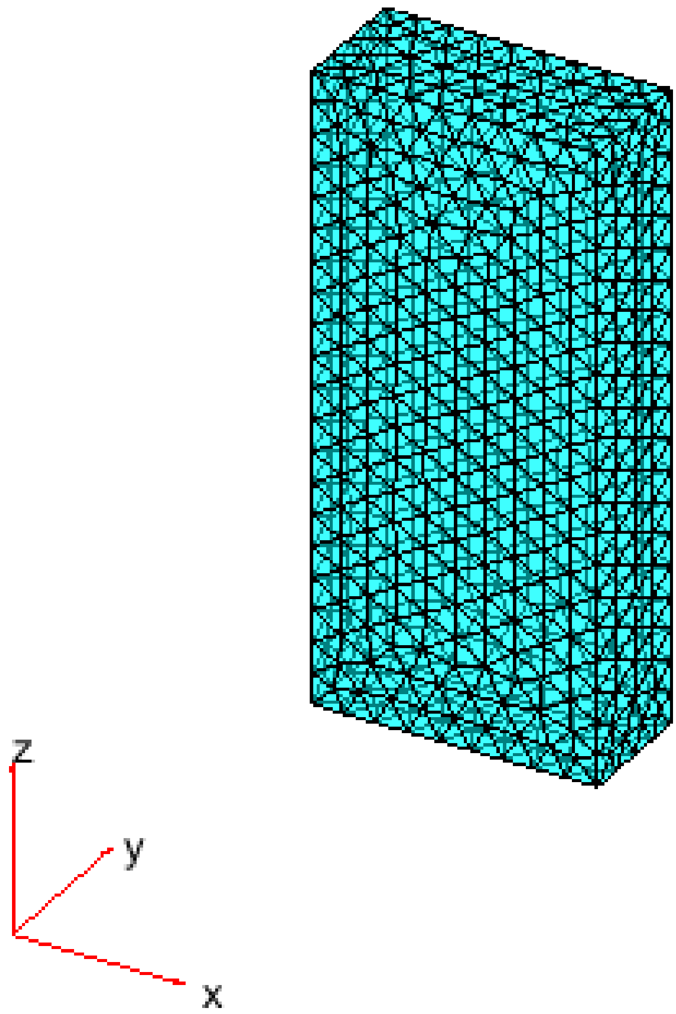


Figure 44 This figure represents the mesh of 0.5m covering the multicuboid represented in figure 33

In this case we consider that the pressure is applied, on the face 2, from the 0 instant to the 0,05 seconds.

Hence, from figure 22 we can say, start time is again 0 seconds, while end time is 0.05 seconds. The results are an increased deceleration of the deck on the ground. The expected result is that displacements and velocity will be higher than before.

Neither the line space of the time is not changed, nor the observation time and the number of frames has not been changed. Hence, still is:

0   0.0375   0.0750   0.1125   0.1500

The first result, for what concerns the displacements is shown in figure 47. It shown the frame 0.0375. For same reason of the previous model we neglect the first frame because it plots the initial conditions thus, everything 0, displacement and velocity. Here we might see that the displacements reach 30 meters, as in the previous simulation. The shear waves should be at 11 meters. It means that their contribute could be found in the blue one zone. Here the displacement shown reach, maximum, 0.05m instead of 2mm of the previous case.

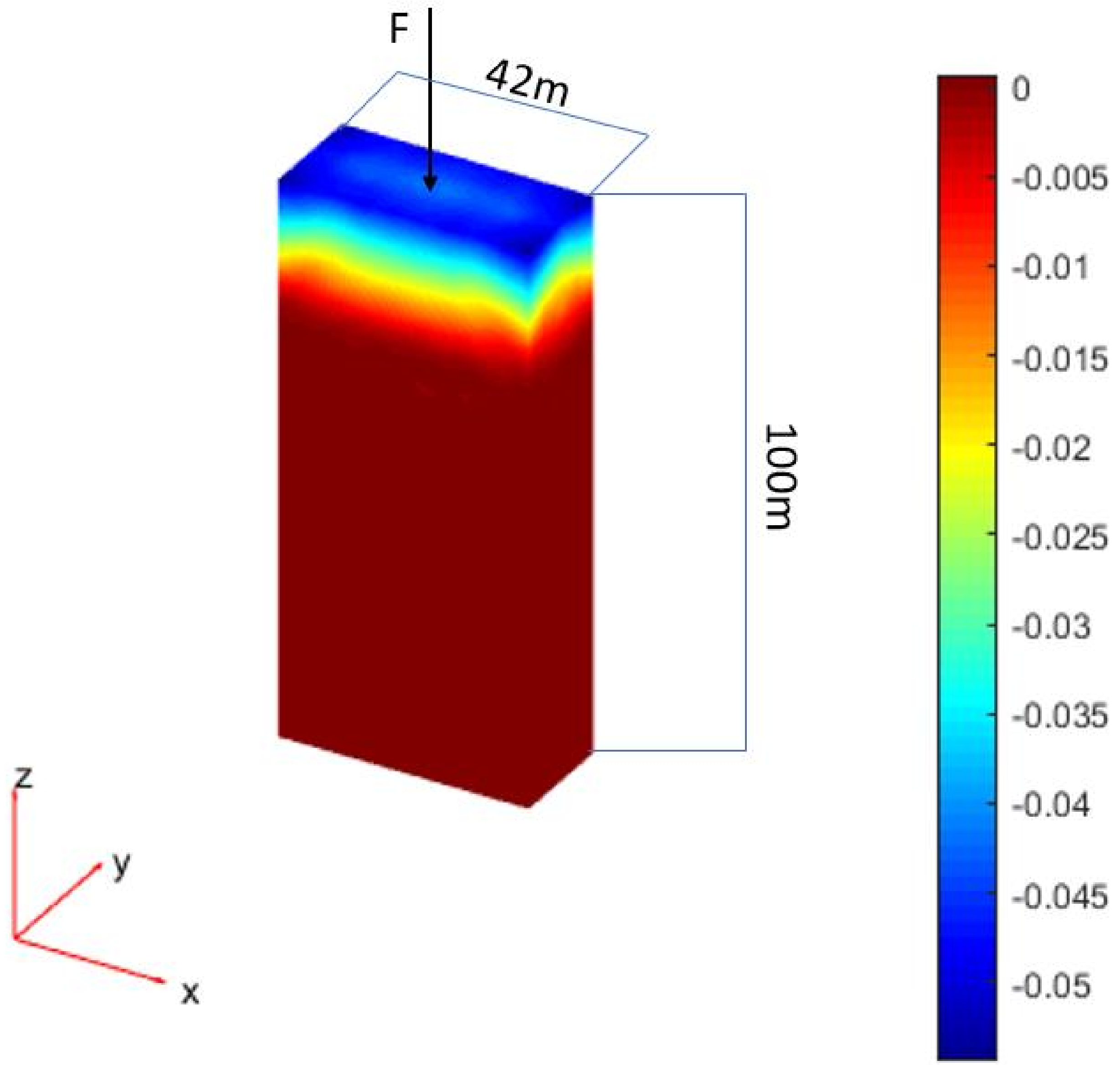


Figure 45 The pressure is applied on the face 2. The scale on the right is in m. This frame is the frame at 0.0375s.

The second frame is shown in figure 46. Here the wave front reaches almost half of the entire length. The compressed zone reaches a displacement of 0.07mm. From the previous frame we might observe an increasing in the maximum displacement. This means the ground is still under compressing strength. The reaction time of the ground is not reached.

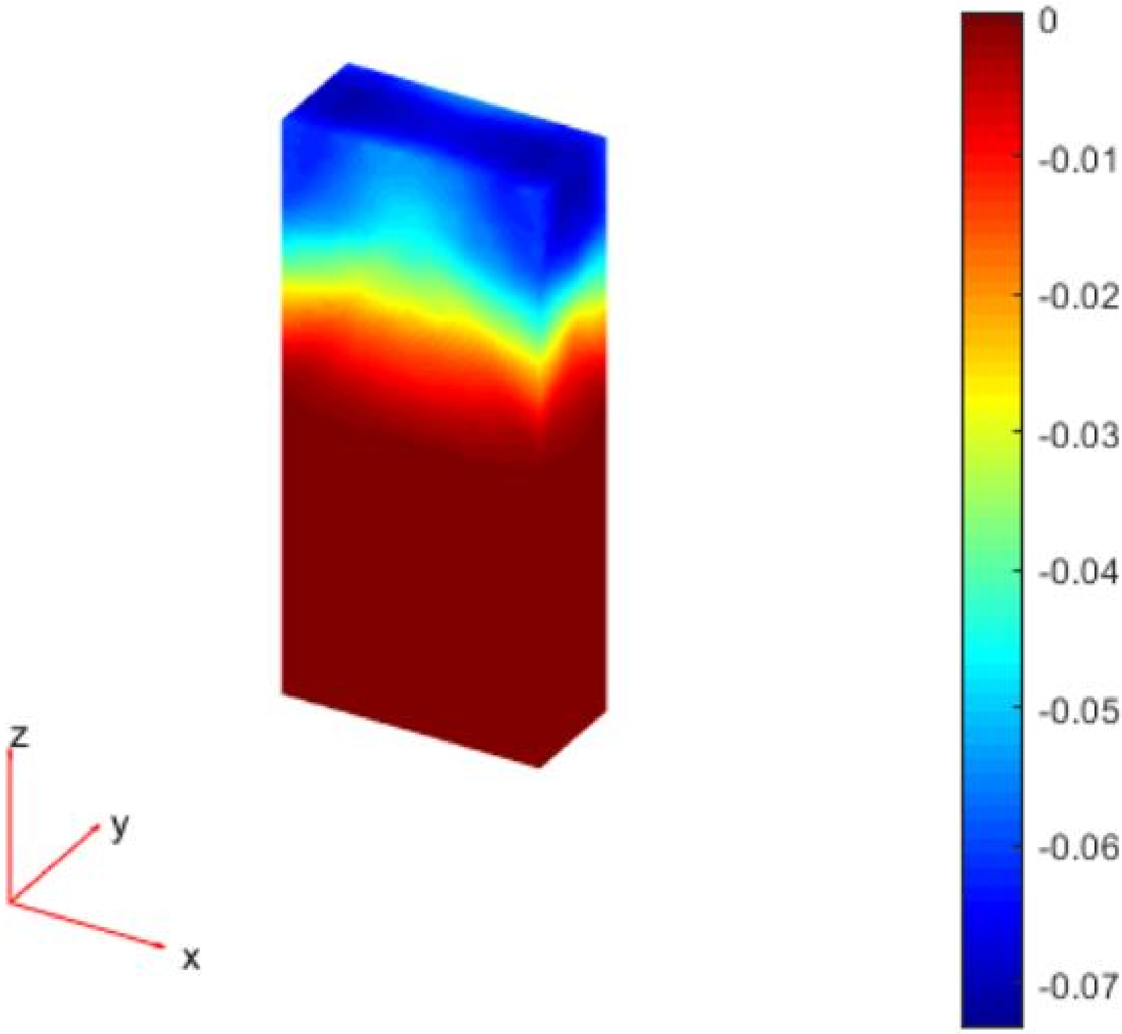


Figure 46 The pressure is applied on the face 2. The scale on the right is in m. This frame is the frame at 0.075s.

In the third frame 70 meters of depth are reached. The total displacement is the same to that found obtained in the second frame. If we compare it with that obtained with the analytical method is possible to see that the order of magnitude is the same. The analytical model says almost 0.04m of permanent displacement. Here, the media is totally elastic and so the total displacement is a little bit higher.

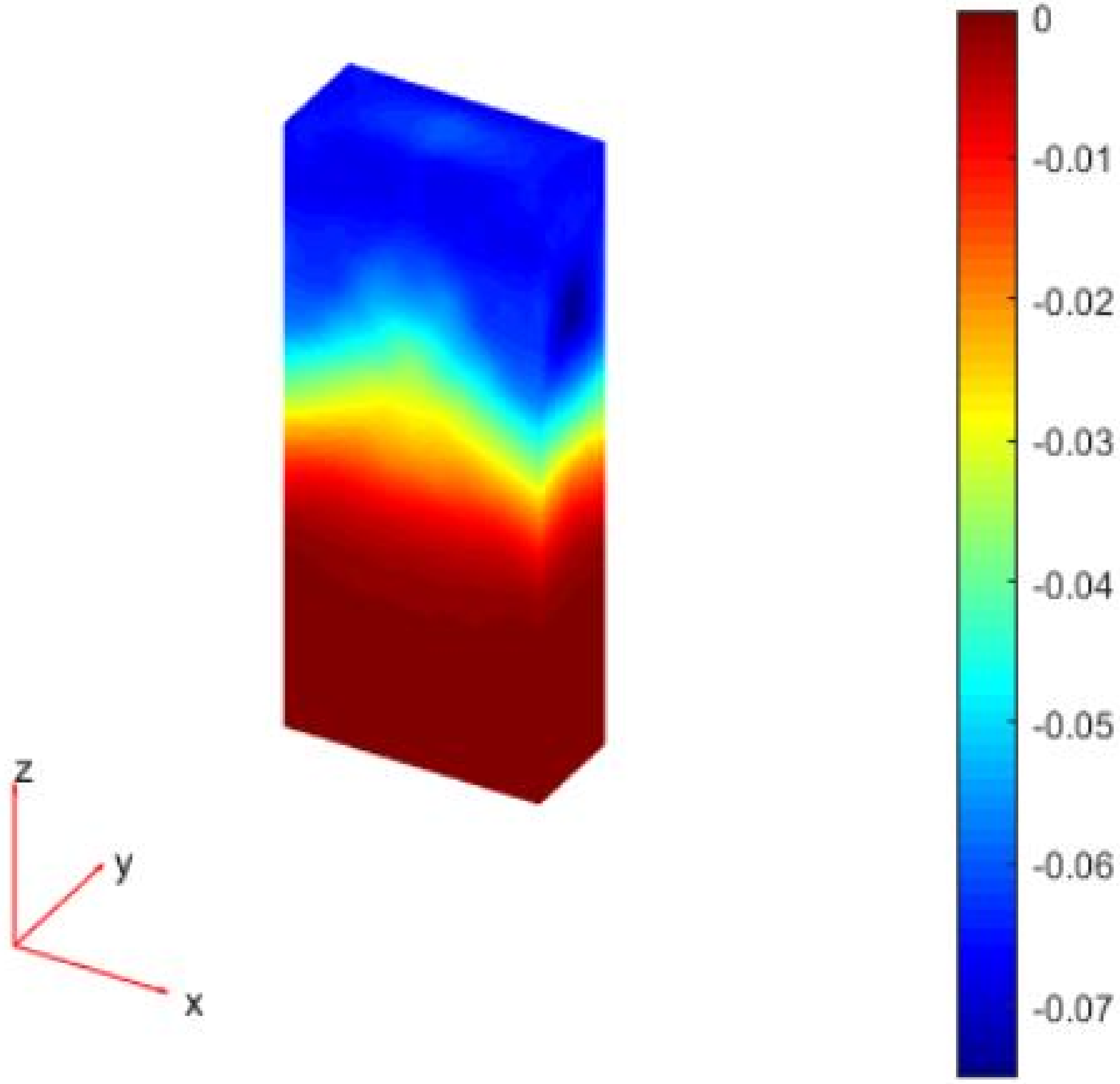


Figure 47 The pressure is applied on the face 2. The scale on the right is in m. This frame is the frame at 0.1125s

In the last frame, shown in figure 48 we have that the wave front reaches almost 100 meters of depth.

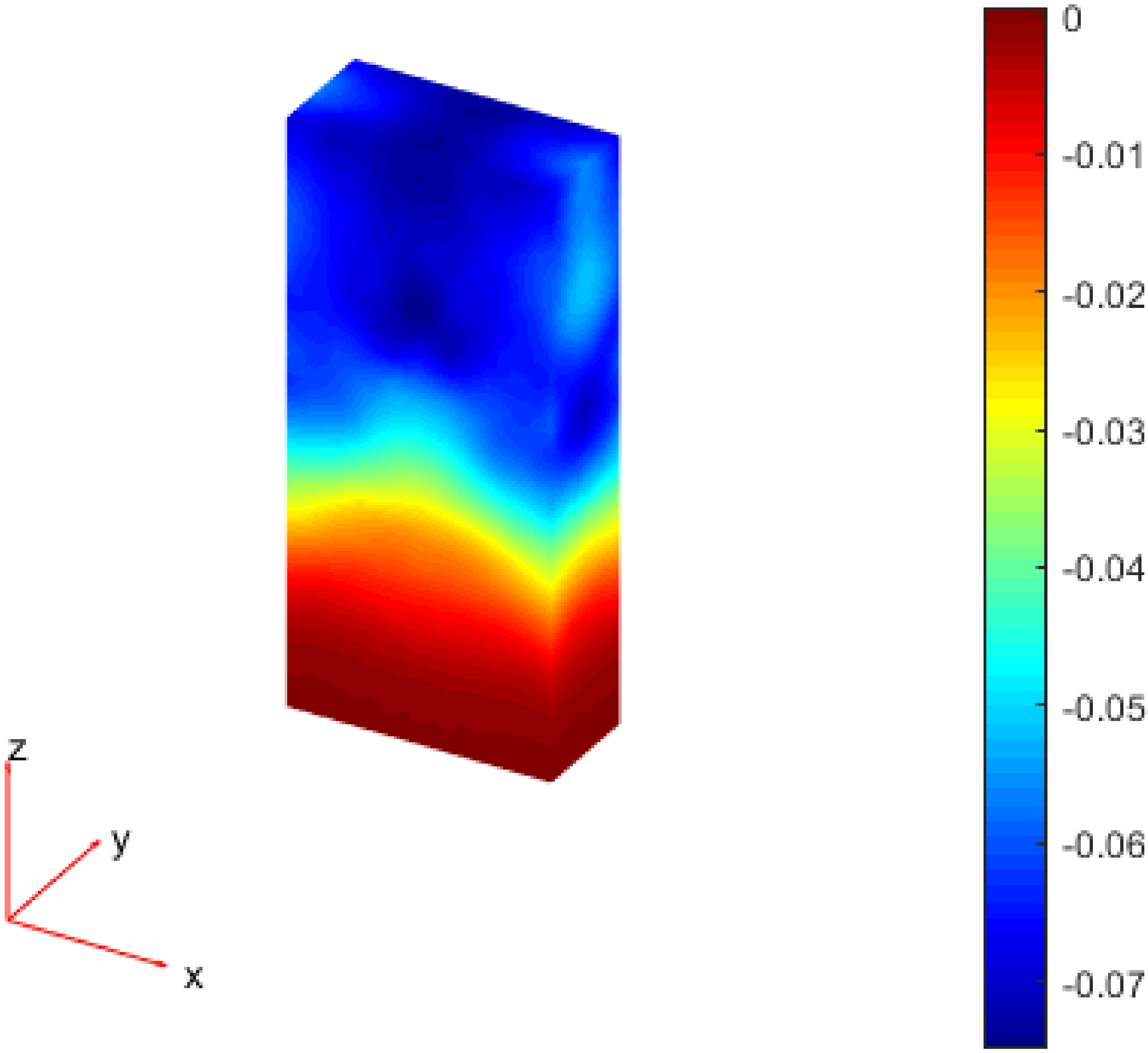


Figure 48 The pressure is applied on the face 2. The scale on the right is in m. This frame is the frame at 0.15s

If we try to represent the frame regarding the velocity we obtain the same result as before. A spotted zone is showed. It does not change also if we increase the frame. The phenomenon could be due to some interferences. It is possible to see it in figure 49.

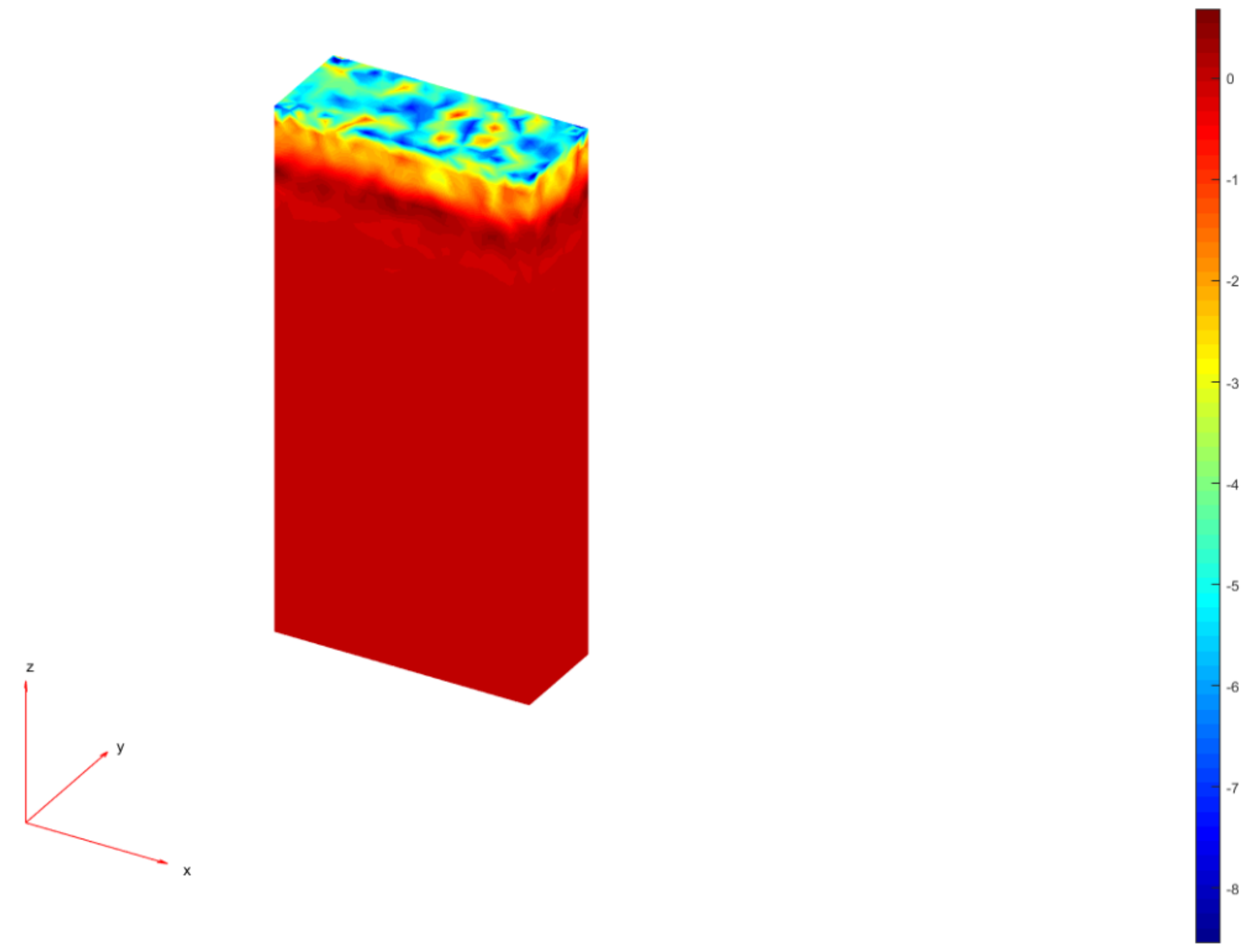


Figure 49 The pressure is applied on the face 2. The scale on the right is in m. This frame is the frame at 0.0375s.

The results obtained until now show a little discrepancy with the analytical model. This could be due to the fact we had developed a pure elastic media. In any case we have the values of vibrations upper than the law limits. It is necessary a system to decrease the value of vibration. We thought to a pillow. The pillow is a stratum of sand. It is considered 5 meters high, 50m long and 20 meters width. To have a simple result we continue to use the 1D model done before. The mechanical properties taken into account for the sand layer are:

The geometry changes as follow in figure 50. It is possible to see the first layer above. It represents the sand. The second layer represent the ground. The pressure is applied over the whole surface 7. It means we are applying the pressure all over the sand. The goal is to adsorb the higher part of the stresses. This goal is needed to preserve the structure all around the impact zone, civil buildings, industrial ones but also the structures underground.

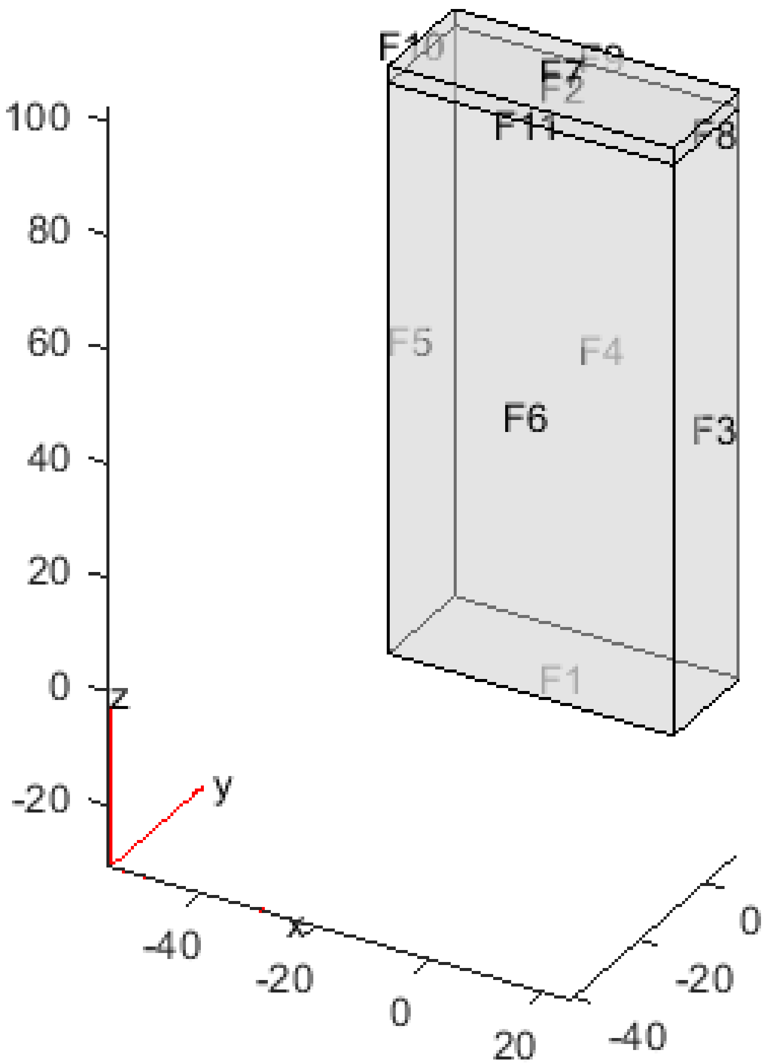


Figure 50 Geometry representig the firt layes wtth an immaginary dept of 100m. The fisrt layer is the sand and is thick 5m. Its width is on the y axes and is equal to 15 meters, the length is on the x axes, equal to 42 meters. Those two length are those of the deck impacting of the face 7.

Also, here the considerations for the mash remain the same. The mesh is 0,5m. 15 times less than the shear wave wavelength.

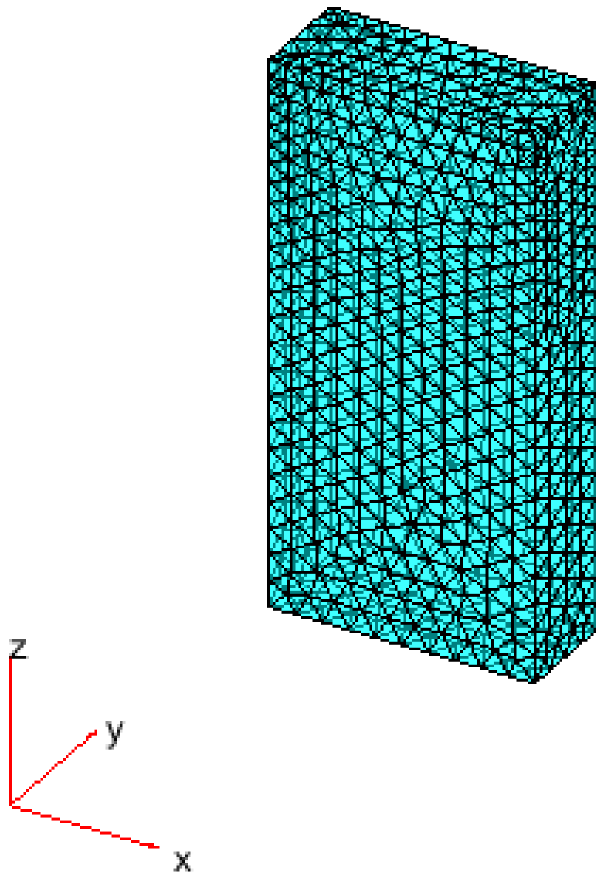


Figure 51 This figure represents the mesh of 0.5m covering the multicuboid represented in figure 33

From figure 52 we might observe that at 0.0375s all the stresses are almost concentrated at the first layer. It means that all the displacement is in the first 5meters. Here we reach a displacement equal to 0.17m almost. Part of the wave front reaches the soils under the sands. Is possible to understand that the energy impacting on the ground now is lower than the case before. In the previous case we had, at 0.0375s, a displacement equal to 0.05m. More than twice that observed now. The waves do not reach completely the ground before. It is in accordance with the physical considerations. Waves go slowly in a less consolidated and less dense material.

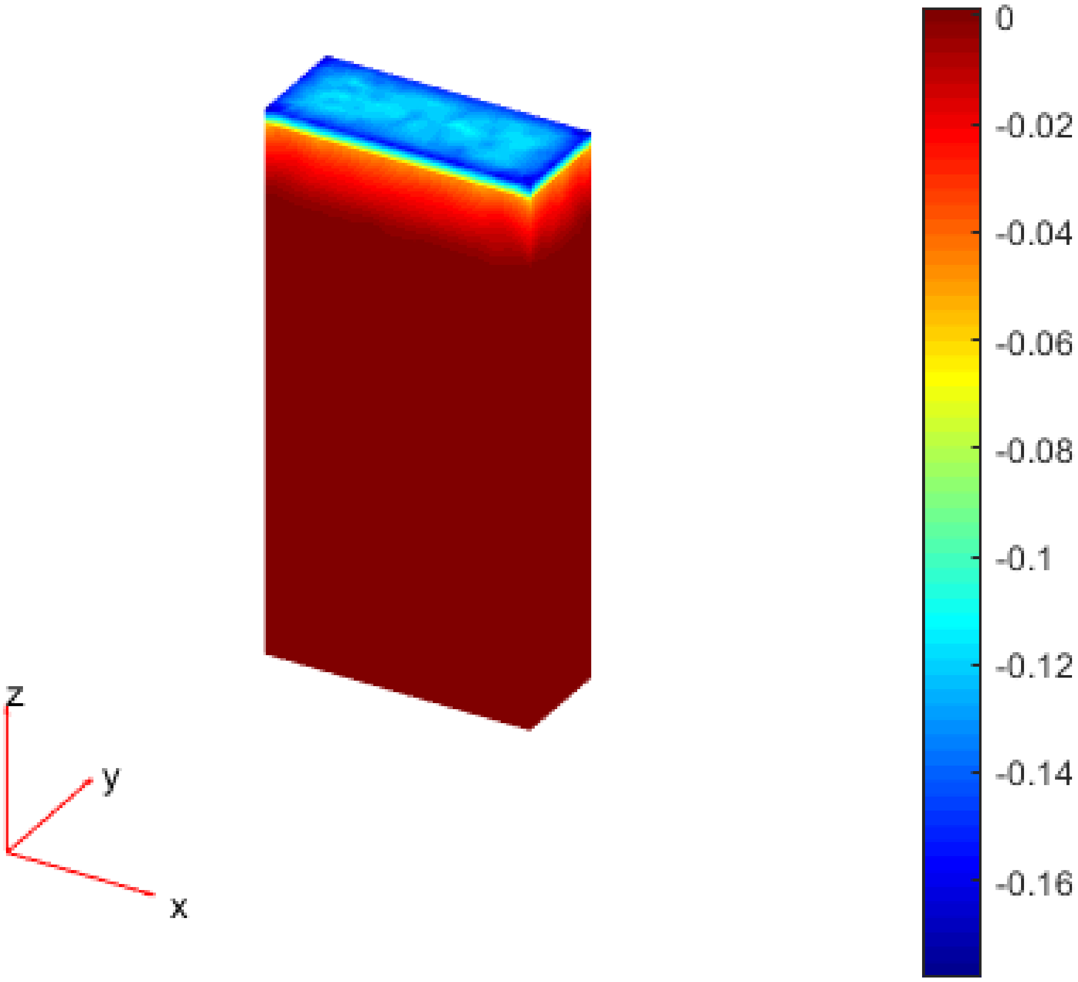


Figure 52 The pressure is applied on the face 2. The scale on the right is in m. This frame is the frame at 0.0375s.

In the second frame, shown in figure 53, is possible to see that the maximum displacement is higher than before. Is necessary to wait until the third frame to understand if it represents the permanent displacement. In the first frame we had maximum 0.16m here, always in the sand interface, we have 0.29m. The wave front now reaches almost the half length of the body. The registered displacement is around 0.05 at the boundary of the wave front and 0.15m at the interface between the sand and the soil.

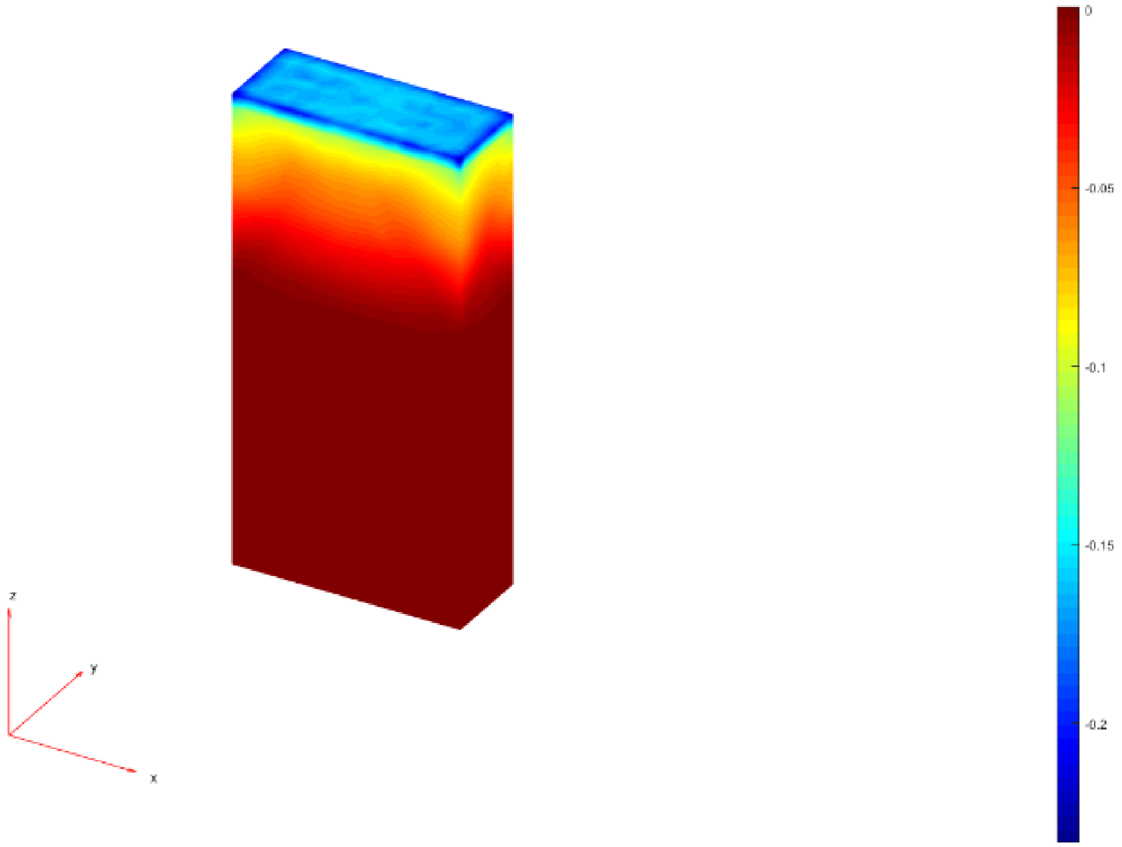


Figure 53 The pressure is applied on the face 7. The scale on the right is in m. This frame is the frame at 0.075s.

In figure 54 is possible to see that the maximum displacement is decreasing. This is due to the elasticity of the model. The wave front do not reach the total depth of the body, as done in the previous simulation. In the soil is possible to see a displacement which reaches 30m with a magnitude of 0.05m. The sand is less compressed, it is passed from 0.29m of displacement to 0.25. If we look to the analytical model, we might observe that we calculated a maximum displacement of 40cm but a permanent displacement of almost 0.2m. In this case our elastic model fits really well with the analytical calculation. Hence, all the conclusion reached with the analytical one could be done also in this case.

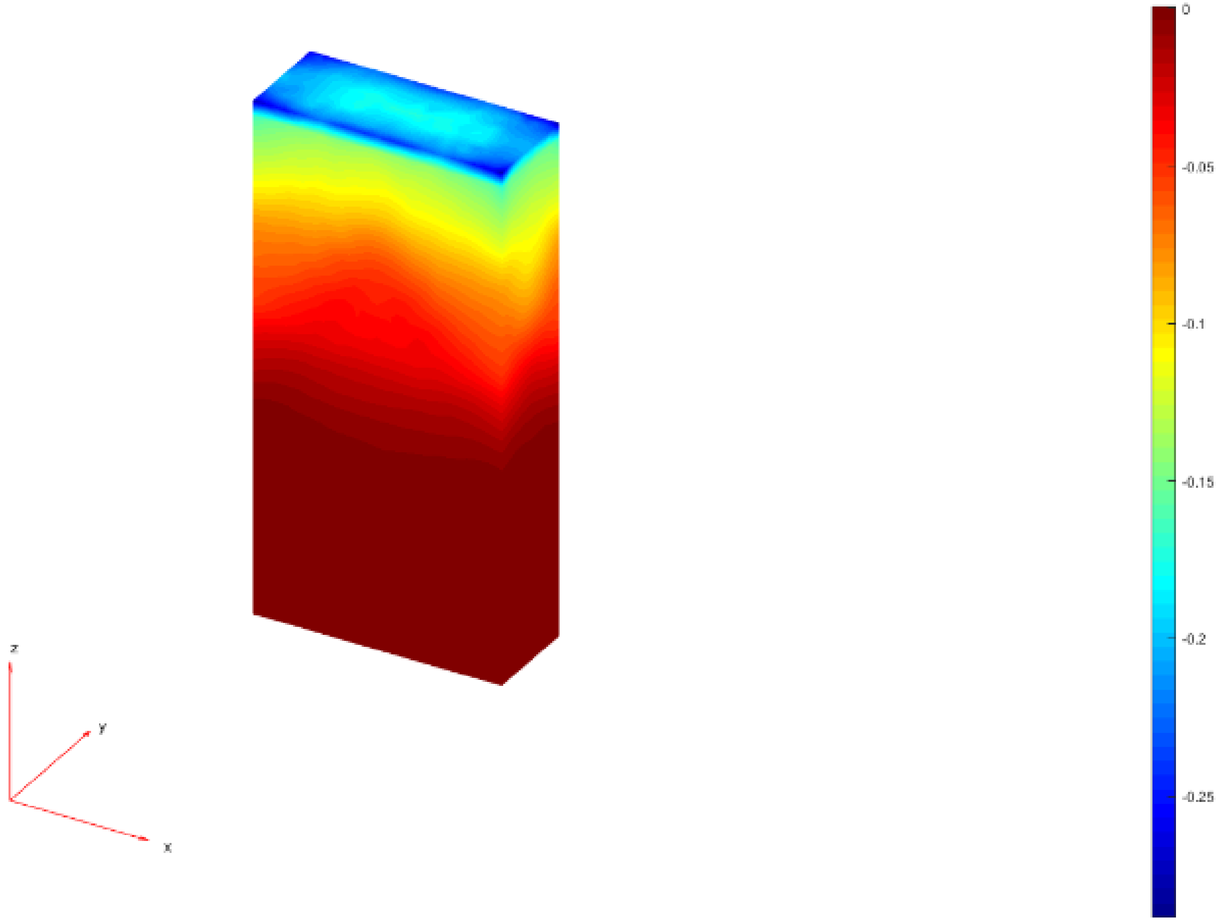


Figure 54 The pressure is applied on the face 2. The scale on the right is in m. This frame is the frame at 0.1125s.

In the last frame, represented in figure 55, we have the following situation: the total displacement in the sand surface reaches 0.3m. In the soil part we might observe a maximum displacement of 0.15m. Those are reached slowly than before and so the effects on the structures are negligible.

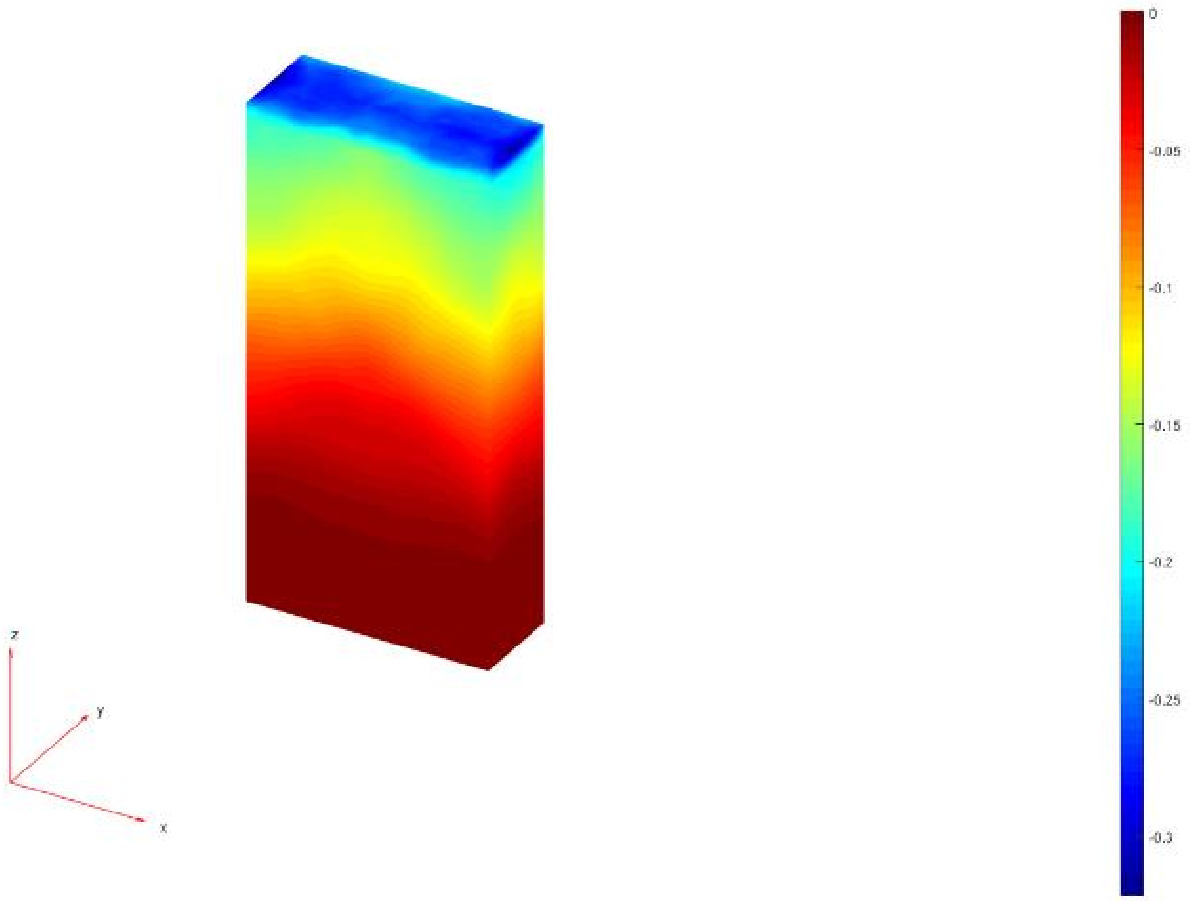


Figure 55 The pressure is applied on the face 7. The scale on the right is in m. This frame is the frame at 0.15s.

The plot of the velocities shown also here the same problem. The wave front reaches at least 20meters. Compared with the case without the sand pillow we might say that the wave front goes slowly than before.

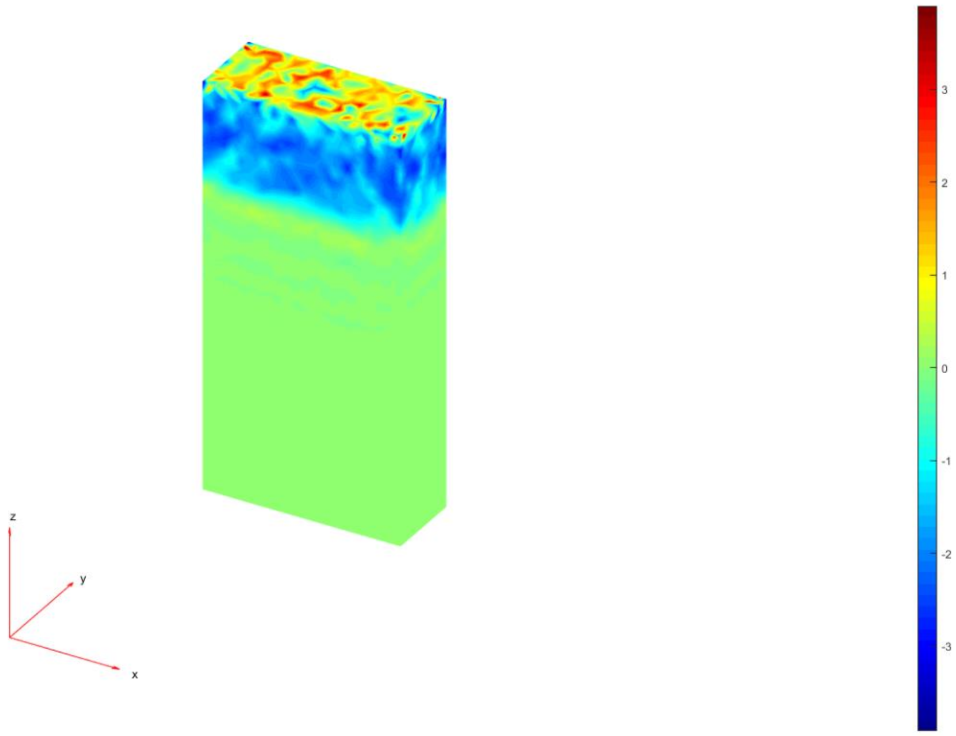


Figure 56 The pressure is applied on the face 7. The scale on the right is in m/s. This frame is the frame at 0.0375s.

Considering the solution with the sand over the ground surface in accordance with the threshold given by the law, we can model the reaction of the underground structures considering the pillow of sand already present. The consideration done in the analytical solution are valid also here. The pressured is considered as an equivalent one. The model is supposed to be in two dimensions and the geometry is simplified as follows in figure 57.

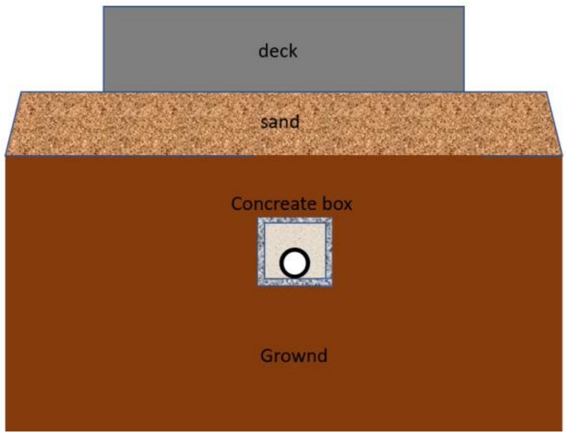


Figure 57Representation of the geometry during the impact moment. Above all is represented the deck impacting over the sand pillow. Below there is the ground with in the concreate box.

We consider the pipeline inside a reinforced concreate box. It is buried under 2 meters of soil. Its dimensions are 2 meters in height and 2 meters in width. It is buried under the first layer. The first layer is supposed to be 50meters deep and 90width. All the geophysical properties of the first layer are applied around the concreate box. Pressure is applied over 30 meters above the concreate box. On the total body is applied the lithological gradient of pressure. For the mesh we consider a 20cm. The Geometry is obtaining by eliminating the box of concreate from the total mass. The program could be found in the appendix. The obtained result is shown in figure 58, her following.

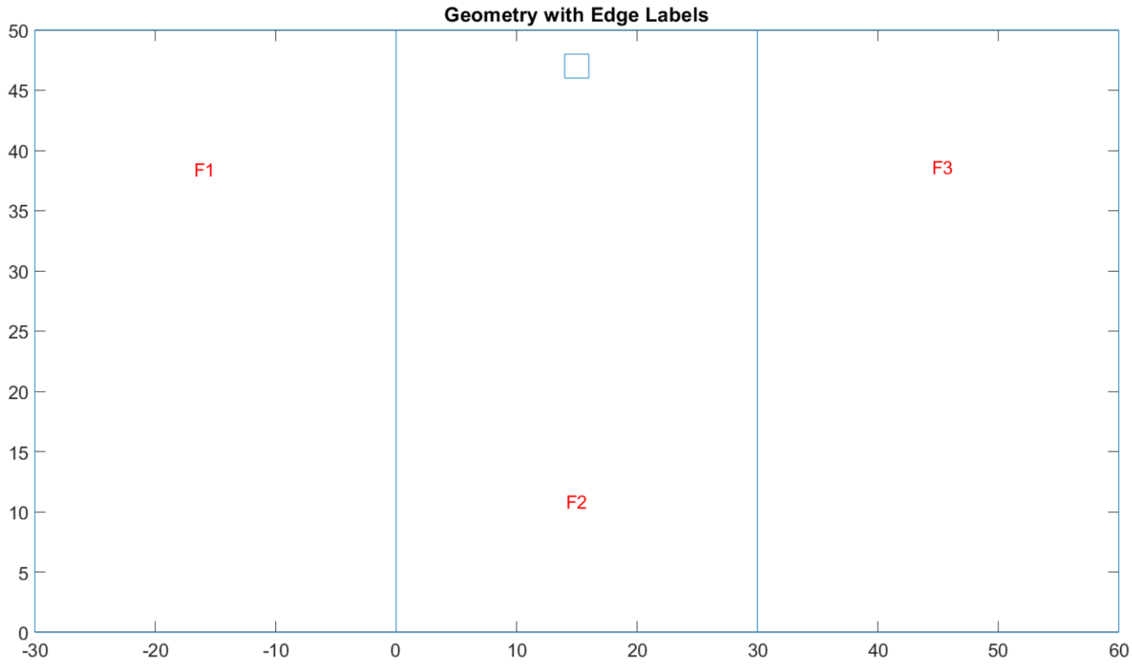


Figure 58It is possible to see that the ground is divided in three zone, F1, F2, F3. In the upper part of F2 is applied the equivalent pressure. Over F1, F2, F3 is applied the lithostatic pressure. In the F2 there is a square of empty material, it represents the concreate box.

The concreate box is represented as an empty material because we need just the result on the surface of it. Also, this model is completely elastic, hence, describes better rocks than soil. The mesh is shown in figure 59:

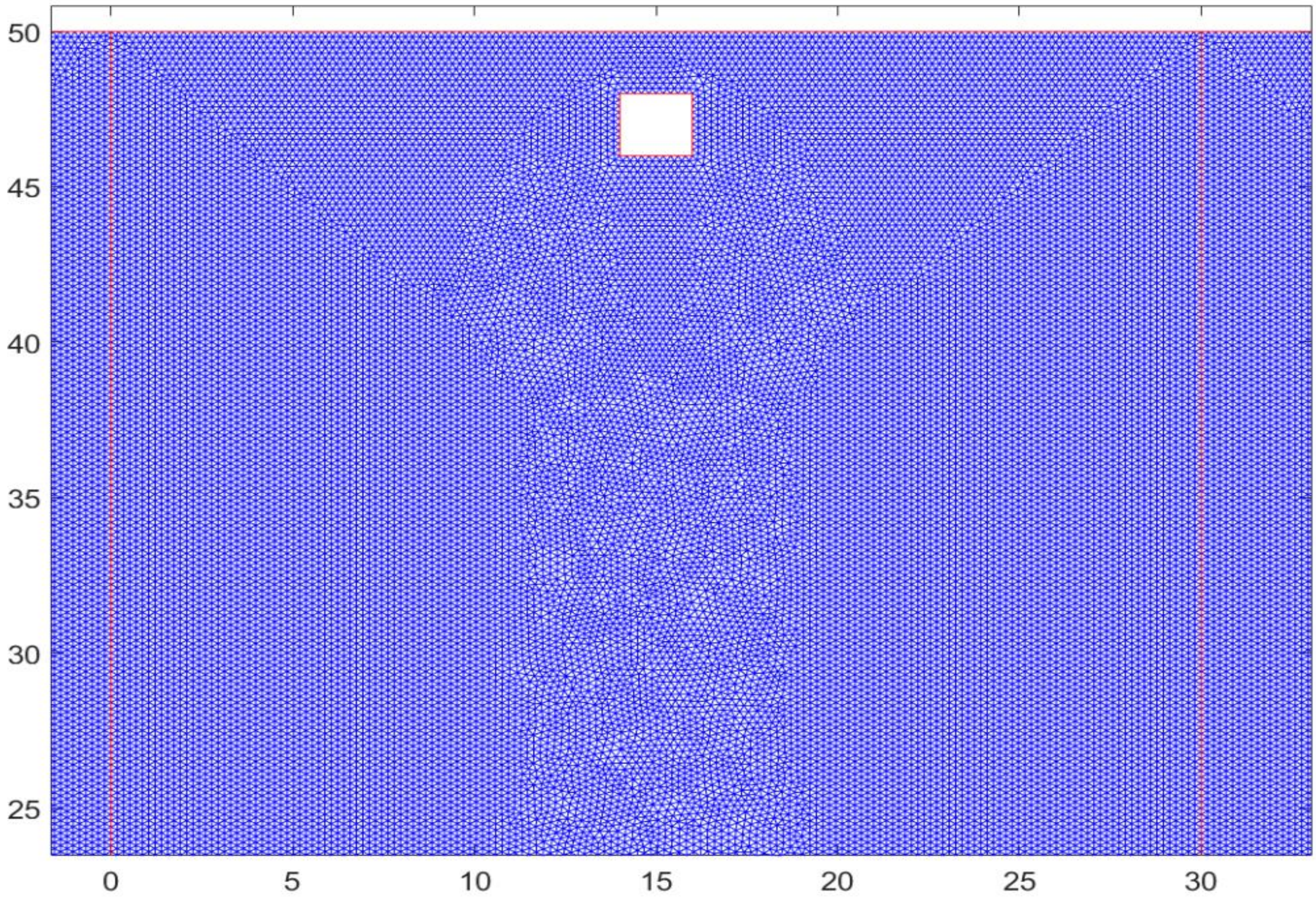


Figure 59The mesh covering all the body. The square in the middle represent the concreates box. It is considered void, that is why the mesh does not cover it. In the picture is represented a zoom on the central part of the plot, just to highlight the texture of the mesh.

The results of the simulation are the stresses around the empty space. In figure 60 and 61 are represented respectively the x and y stresses.

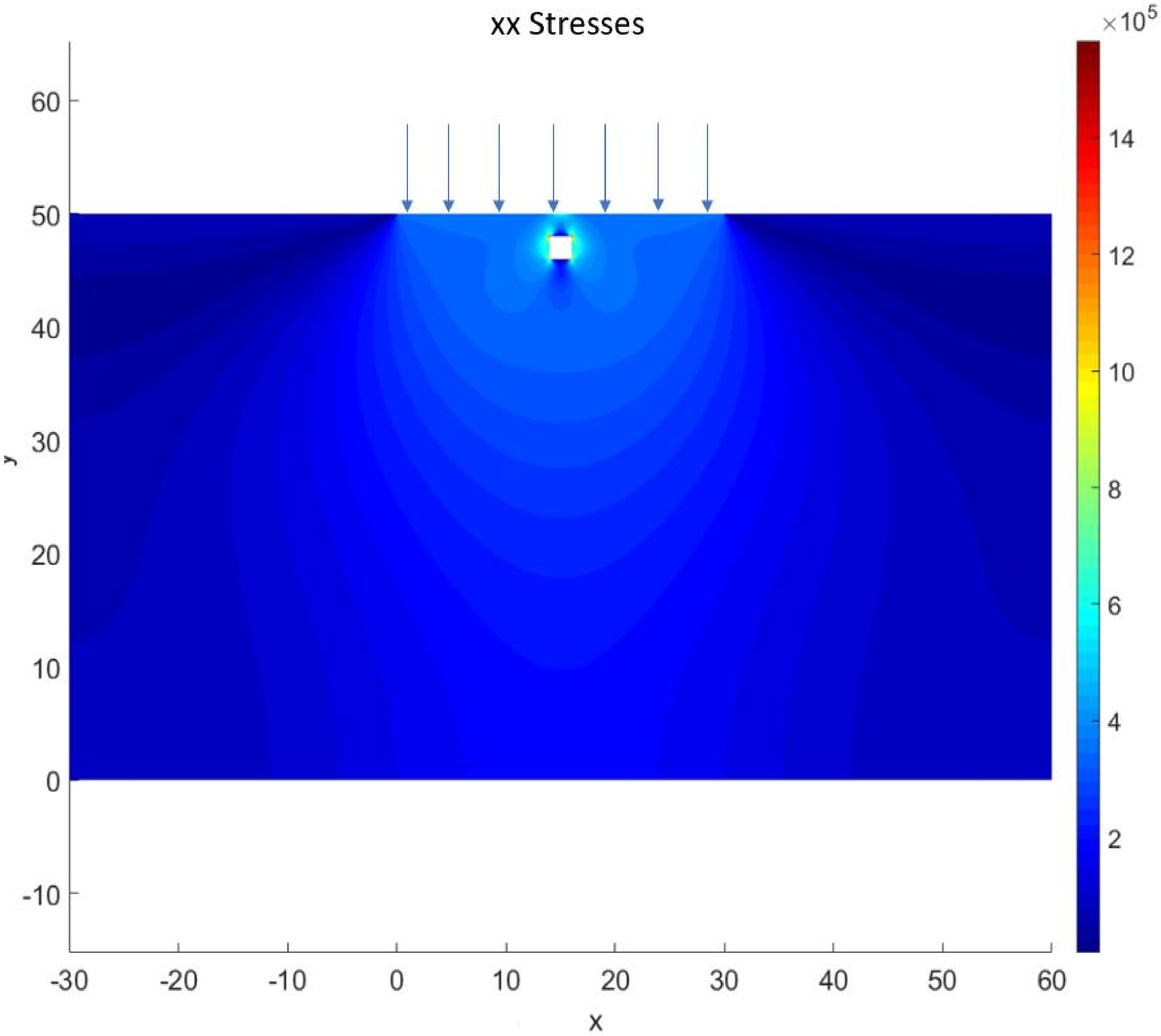


Figure 60Are represented xx stresses. The figure is represented in Pa, on the right size is possible to see the scale.

The application zone of the pressure is highlight by the typical propagation zone, the bulb shape. The stress all around the body is the lithostatic one. In the ground we reach a maximum value of 5-6bars. Around the reinforced concreate boxes are reached the maximum stresses. The same consideration could be done for the plot representing the stresses in y axes. To better understand the values of the stressed we need a zoom over the concreate boxes.

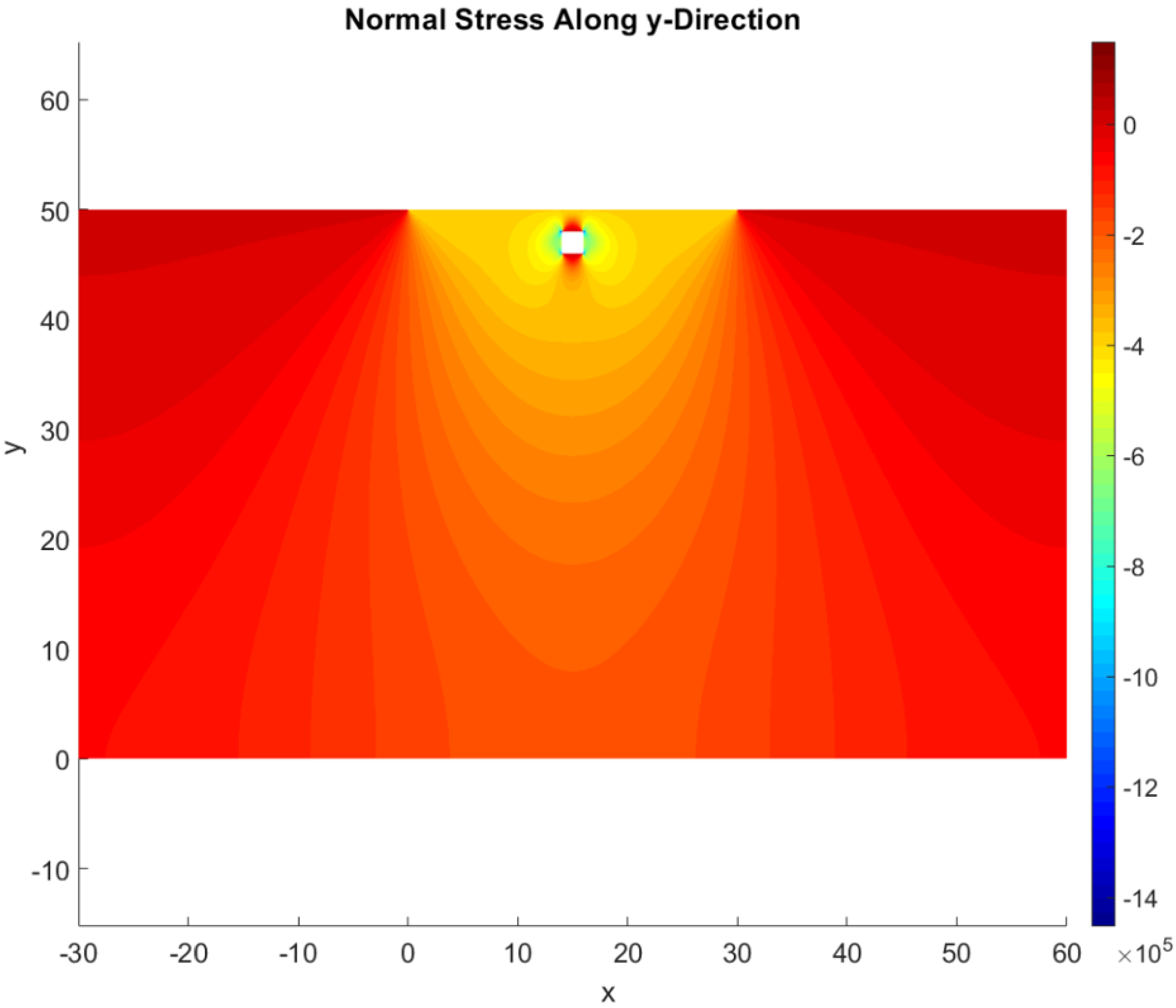


Figure 61 Here are represented the stresses along x. The figure is represented in Pa, on the right size is possible to see the scale...

The zoom on the reinforced concreate is shown in figure 62. To be more conservative we represent the stresses with the maximum magnitude. From the plot 60 and 61 it is possible to see we have same values, that means the distribution is isotropic on the x and y direction. So, does not change nothing for the zoom. The scale used for picture 62 is the same as that used in figure 61 and 60. In particular the picture represents the zoom of the stresses along x axes.

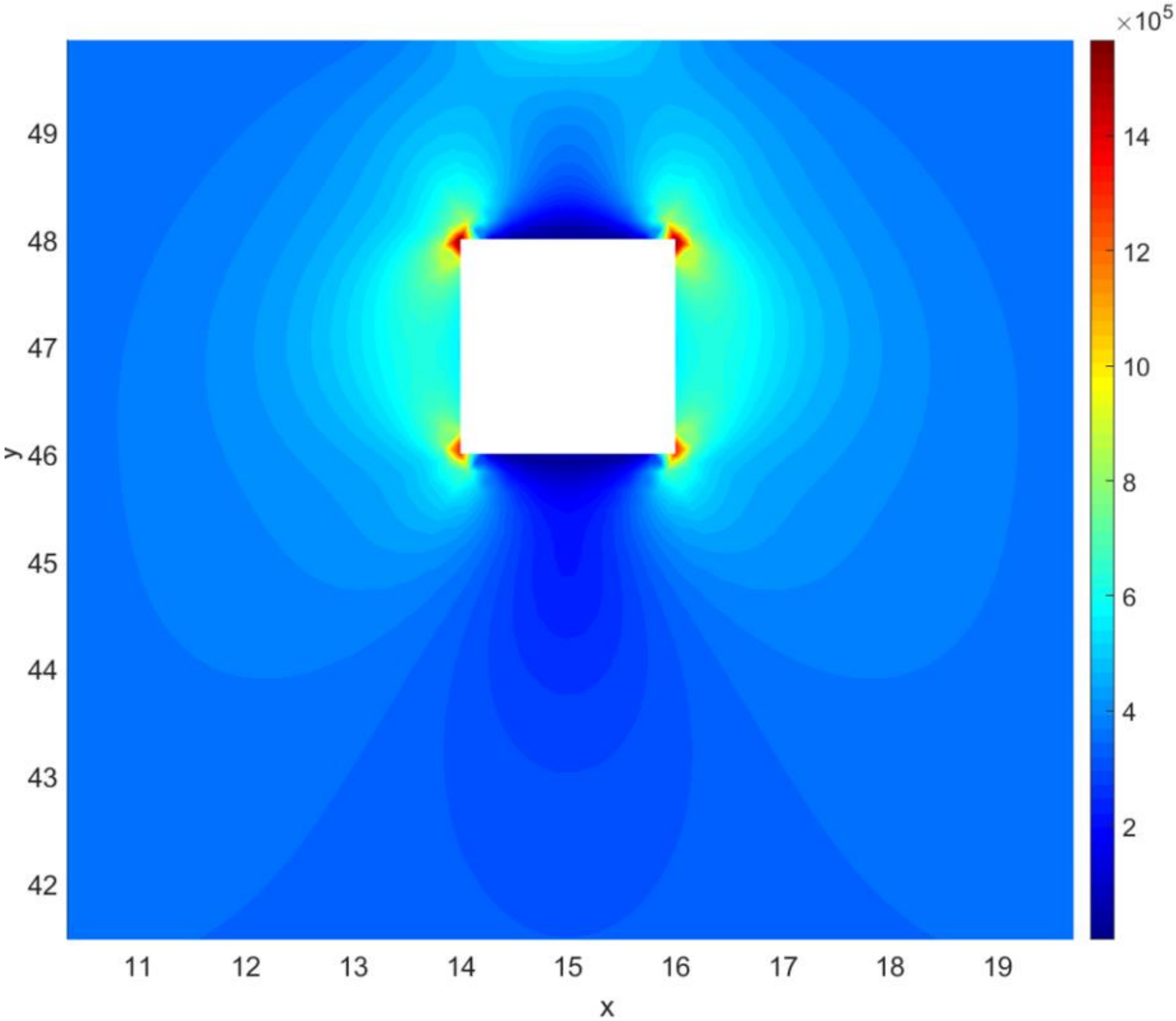


Figure 62 Here are represented the zoom of the stresses along. The figure is represented in Pa, on the right size is possible to see the scale.

The maximum values of the stresses are reached on the wedge of the box. That reaches almost 1.5MPa. The high stresses zone propagates for 1,5 meters far from the box, and it is represented by di light blue zone. The upper and the lower part of the box are interested by the lower value of the stresses, they are almost equal to the lithostatic pressure. In any cases the obtained values do not reach the project limit of the reinforced concreate.

Our considerations about the model have to take into account that the physics of the analysis is completely elastic. The real problem cannot be totally elastic. The physic of the problem is viscoelastic; hence we must change simulator, in order to better approximate the problem. The program the program that can compute the viscoelasticity is Flak. Flak has a tool called Geomechanics. With it is possible to consider the viscous dissipation at the ground. Another advantage of Flak is it is possible to model the force as a gaussian impulse.

It is possible to see all the codes used to perform the simulation with matlab in APPENDIX C.

Viscoelasticity of the ground may be modelled by the Rayleigh approach. Thanks to the Rayleigh model the dumping matrix, C, is defined as a linear combination of mass (M) and stiffness (K) matrixes:

$$C = \alpha_R M + \beta_R K$$

The coefficients α_R and β_R have been calculated through the double frequency method, (Lanzo 2004) as follow:

$$\alpha_R = \zeta_0 \frac{2\omega_i \omega_j}{\omega_i + \omega_j}$$

$$\beta_R = \zeta_0 \frac{2}{\omega_i + \omega_j}$$

Where ζ_0, is the viscous dumping ratio and ω_i and $\omega_j$are the control frequencies. The dumping ratio is obtained by this formula:

$$\zeta_0 = \frac{c}{c_z} = \frac{c}{2\sqrt{km}} = \frac{c}{2m\omega_0}$$

Where appear c, the dumping coefficient, c_z which is the critical dumping coefficient and the ω_0 that is defined ad the natural pulse or circular frequency of the oscillator, in this case the first layer of the ground. It is obtained:

$$\omega_0 = \frac{2\pi}{T_0}$$

By imposing the natural or own period of the oscillator, T_0, equal to:

$$T_0 = 2\pi \sqrt{\frac{m}{k}}$$

Here we can see other two terms, the m and the k. They are respectively the mass of the impact body and the stiffness of the spring-dumper model. Through the Rayleigh coefficient t is possible to retrieve the dumping ratio for every frequency:

$$\zeta_k = \alpha_R \frac{1}{2\omega_k} + \beta_R \frac{\omega_k}{2}$$

From this function it is possible to draw a plot $\zeta_k$vs ω_k, the result is in the following figure:

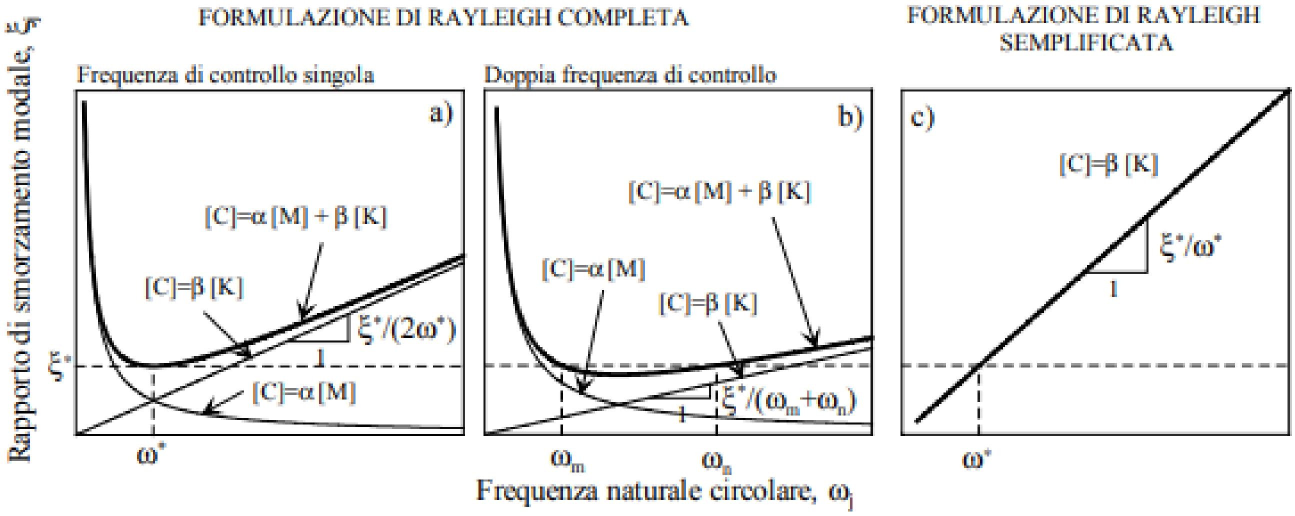


Figure 63

Once drawn the plots will be possible to know and estimate the Rayleigh coefficients, α_R and β_R. This would be true for every frequency and dumping coefficient. The Rayleigh coefficient obtained graphically, for different depth, and for three different natural frequencies, with dumping equal to 0.05 are resumed in table 8.

Table 8: Values of α_R and β_R for dumping equal to 0.05 and relative frequencies considered at three different seismic events.

		A-TMZ000		A-STU270		E-NCB090	
layer #	*z* [m]	α_R	β_R	α_R	β_R	α_R	β_R
1	0-1	0.516	0.0036	0.344	0.0073	0.619	0.0015
2	1-4	0.674	0.0028	0.449	0.0056	0.786	0.0014
3	4-6	0.750	0.0025	0.500	0.0050	0.875	0.0013
4	6-10	0.879	0.0021	0.586	0.0043	0.977	0.0014
5	10-20	0.711	0.0035	0.680	0.0037	1.133	0.0012
6	20-30	0.772	0.0032	0.772	0.0032	1.286	0.0011

The three seismic events in consideration are A-TMZ000, A-STU270 and E-NCB090. Their natural frequenciy are plotted in figure 64, 65 and 66.

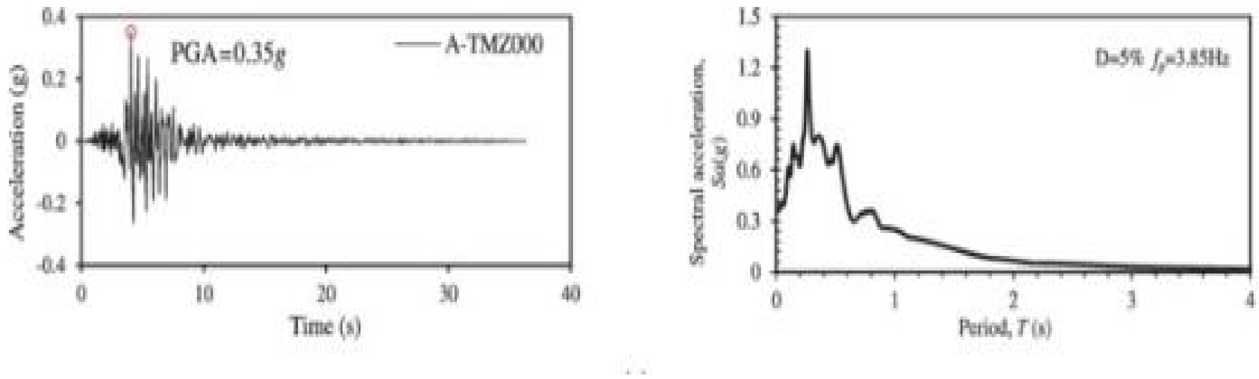


Figure 64Acceleration time histories and response spectra: A-TMZ000 Assessments of Kinematic Bending Moment at Pile Head in Seismic Area December 2018 Journal of Earthquake Engineering

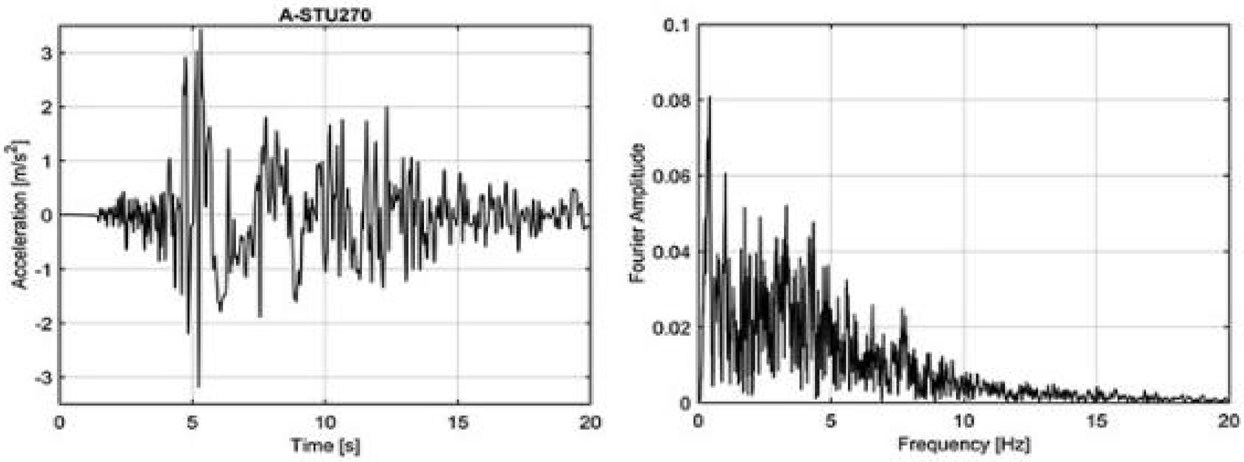


Figure 65Acceleration time history (left) and Fourier spectrum (right), A-STU270 (scaled at 0.35 g).

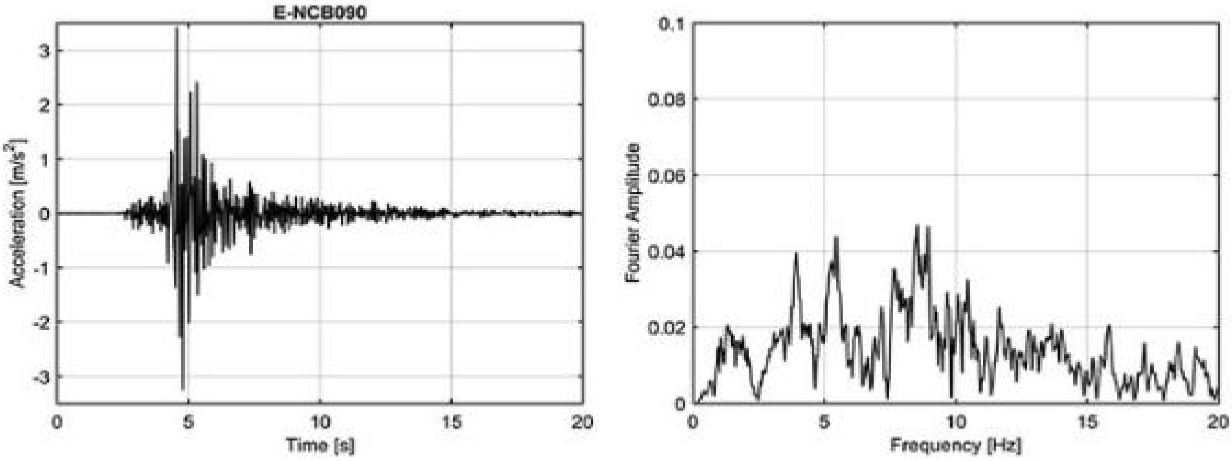


Figure 66KIN SP: a Boundary Element Method based code for single pile kinematic bending in layered soil February 2018 Journal of Rock Mechanics and Geotechnical Engineering 10(1)

The frequencies used for the estimation of the Rayleigh coefficient are the most dominant in the accelerometer seen in the figure above.

The dimension of the model done in flak are the same to those used in matlab. The only one difference in terms of geometry is the presence con the box's thick. We insert it to simulate the response of the concreate to the stresses. The boundary conditions are supposed the same as before. The lithostatic pressure, the roller around the perimeter of the geometry and the pressure applied as an equivalent one. The differences are made for the geophysical properties of the ground. We insert the Rayleigh coefficient, the attrition angle of the soil and the coherence of the material

The obtained result is the following, shown in figure 67:

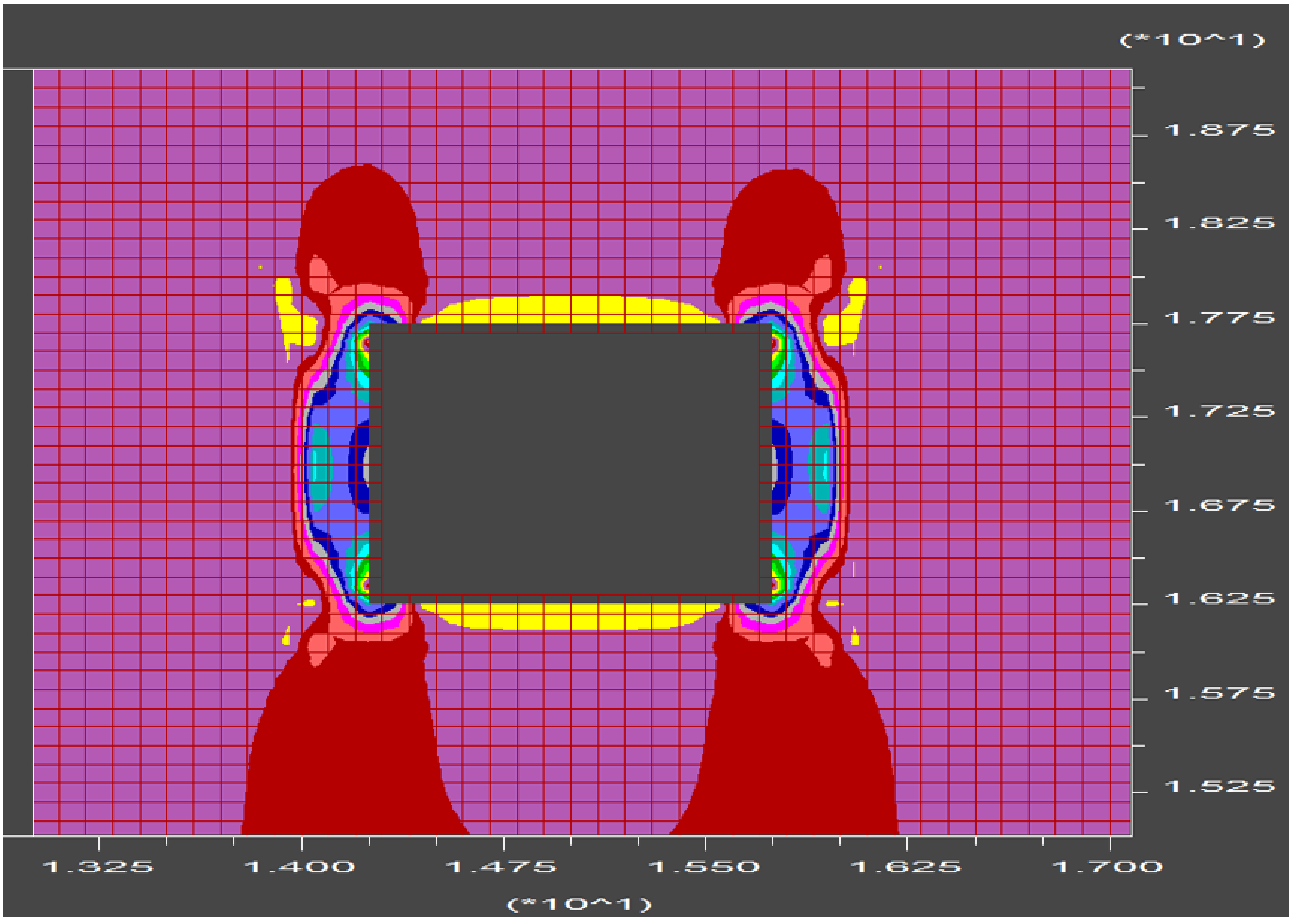


Figure 67 Here are represented the stresses along x. The values reported on x and y axes have to been intended as meters and should be multiplied for 0.1. The 0 of the axes has been considered on the lower left vertex of the system, not present in the figure. To better understand look at picture 57.

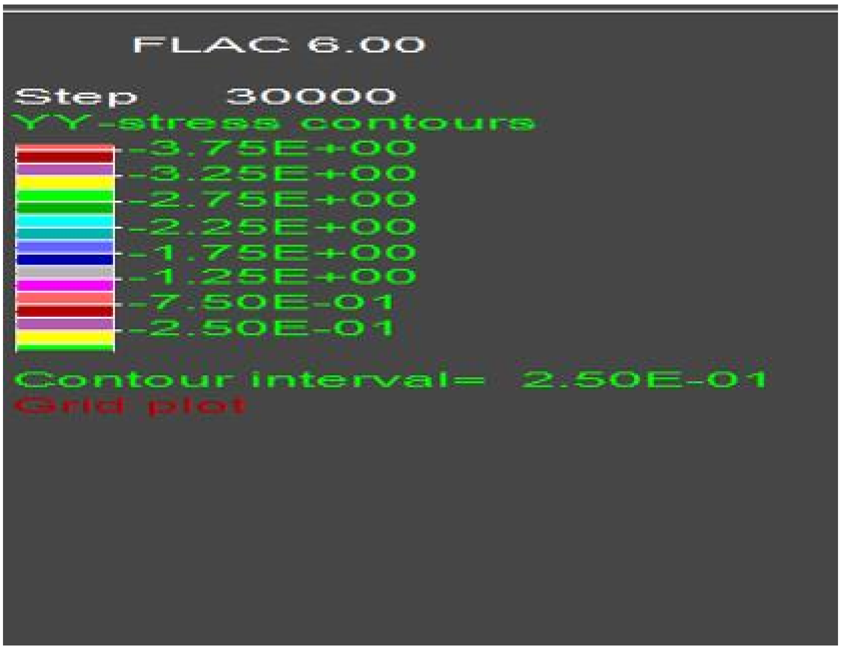


Figure 68This is the scale of colors represented in figure 66. The highest value is the light pink one and the lower the light green. The unit value is MPa.

The result of the plastic simulation shown higher values of stresses than those performed with the elastic model. In any cases the order of magnitude is the same. Also, the distribution of the stresses around the boxes are similar. Here the highest values are reached around the wedges, as before. The values are almost 3.75Mpa, twice the values obtained before. It still remains under the project value of us concreate. To better appreciate the stresses change, over the reinforced concreate upper surface, before and after the application of the load we might plot them.

The result is shown in figure 69. The values will be plotted in absolute value; hence they will look like traction values instead compression values as they are.

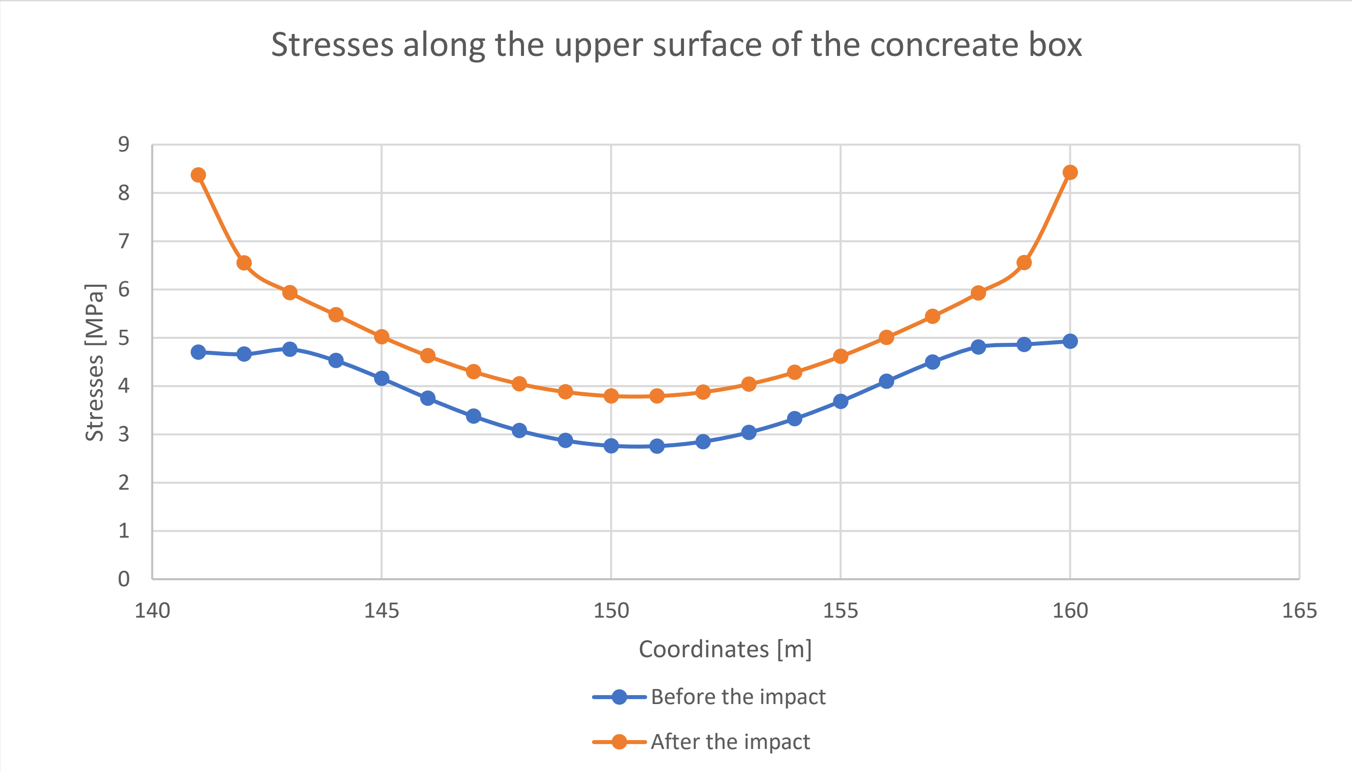


Figure 69In the picture are represented the stress along the upper surface of the reinforced concreate box before and after the deck impact. The orange line represents the values after the impact while the blue one the values before the impact.

From picture 68 is possible to appreciate the behavior of the stressed before and after the deck impact on the ground. The most interesting thing is the change of the behavior on the wedges of the box. Before the impact the maximum value of the stresses is around 4.8MPa, the minimum is around 3MPa. In the middle part of the box there is an increasing of the average value (before and after the impact) of just a mega pascal. On the wedges we pass from 4.8MPa to more than 8MPa. The difference is almost 4 times bigger than the average behavior on the middle. It is possible to see that the wedges are the most solicited part of the box. In any case, with the concreate chosen for the construction of the box we are still in safety condition.

4) Conclusions

The porpoise of this thesis was to carry out the study on vibrations propagation produced by the impact on the ground of a pillar demolished with explosives.

The characterization of the problem physic was a fundamental part of the analysis. Understand all the geophysical parameter in place, has been necessary to develop the model. Geophysical characterization has principally done empirically with in situ measurements, especially for what concerns the wave propagation velocity, for both, pressure and shear waves. Elastic parameters have been determined analytically starting from the empirical measurements. The characterization of the pillar's components, hence, the deck and the pole were, firstly, performed analytically, then to enhance the accuracy of the data we performed it with the support of a CAD. Once calculated the volume and the mass of the pillar we compare it with the existing estimation of the real data, to check if there are discrepancies. The last part on the characterization of the problem was to calculate the energies in place. To evaluate the energy at the impact we simply considered the pillar as a grave in free falling, then all the kinetic energy is converted. It is converted in three parts. Dissipation as heat and sound, elastic propagation of the waves and viscous permanent deformation of the ground.

Our purpose was to understand the amount of energy transformed in vibrations, its different components and compare this amount with the threshold for the surround structures, aboveground and underground. To model the propagation of the waves in the ground we used two method. The first analytical and the second was the numerical modelling. In the first approach we approximate the problem with the well-known Kevin-Voigt spring-dumper model. It consists in the simulation of the impact and the response of the ground with a second order differential equation, forced and dumped. The terms of the equations represent the elastic part, the permanent deformation and the oscillation and the forces imposed by the action body. To solve the model, we had to hypnotize the initial conditions. Those initial conditions must be valid for both the analytical and the numerical model, in this way we have been coherent we the obtained results. The initial conditions we took are the zero velocity and zero displacement at time zero, plus the initial force, converted into a pressure, considered as a gaussian distribution. The model highlights a discrepancy among the vibration values obtained and those permitted by the law, more than 30mm/s instead of maximum 8mm/s. To decrease those values, we add a stratum made by loose material, actually, sand. This expedient permitted to obtain values below the threshold law. For what regards the underground works, the analytical approach has been roughly. It has the only purpose to understand the possible order of magnitude of the stress on the buried structures. The chosen approach is easily the Kirsh solution for an anisotropic distribution of the underground stresses. The obtained values are below the project limits typical of the reinforced concreate.

The second part is the creation of the numerical model. It has the scope the confirm and enhance the results obtained with the analytical model. It has been important especially for the buried pipeline. Our numerical model is performed with same initial and boundary condition found for the analytical one. The first approach to the numerical model is completely done in elastic field. The development of the geometry is carried out in different steps. Firstly, we simulated a three layers geometry, exactly tracing the real model. Then, to better understand the propagation phenomenon we simulated just one layer, the first, considering the geometry bidimensional. To obtain a refined result we used a mesh 15 times thinner than the shear wave wavelength. We compared the results of the model, in terms of vibrations and displacements, with the analytical model and with the calculation of the wave frond done preliminary. Both the analytical model and the numerical model are in accordance. Thus, we simulated the sands layer with the numerical model, to compare this solution also. It is evident that the highest part of the energy is adsorbed by the sand layer. the displacements retrieved for the sand have an order of magnitude of 35 cm while for the ground we are around millimeters. It is in accordance with the analytical model. To move through the viscoelastic model, we had to introduce the Rayleigh coefficients. They represent the viscous dissipation involved in the ground. The evaluation of the Rayleigh coefficient is performed with the Lanzo method. Thanks to it we could assume the beta factor negligible compared to the alpha factor. Thus, we had only an elastic matrix. Our model for the buried pipeline is developed considering the sand pillow already in place. The total load on the ground is considered as an equivalent pressure, it is composed by the weight of the sand, the impact pressure and the lithostatic gradient. We developed firstly in completely elastic material, the difference with Kirsh solution wav we could simulate a squared hole instead a circular one. The obtained results were comparable with those obtained by Kirsh solution. Another highlight from the numerical model was the concentration of the stresses on the wedge of the concreate square. Those value are around 1.4Mpa and so below the rupture limit of the reinforced concreate for both, compression and traction. This model is done with the structural mechanics tool of matlab, hence has to be considered in elastic media, so in a rock. To evaluate also the plasticization of the ground, the thick of the reinforced concreate and the properties of the soil, instead those of a rock, we moved thorough a viscoplastoelastic model. It is performed with Flac. here we had to add all the geomechanical properties ad attrition angles and cohesion. The results are of the same order of magnitude but if we look on the wedge of the reinforced concrete it is possible to appreciate that the values are around 3,8MPa, the double of the elastic model. In conclusion it is possible to say that the preliminary evaluation of the vibrations into the ground, without the presence of the sands, go above the threshold limits permitted by the law. It is advised the presence of the sand to control and dissipate the bigger part of the vibration, in this way is possible to preserve above and underground constructions.

5) References

Determining Rayleigh damping parameters of soils for finite element analysis Shen-Haw Ju ,Sheng-Huoo Ni (2007)

Dinamica delle strutture di Giuseppe Muscolino (2001)

Ground improvement G.L Sivakumar babu

Influence of damping and soil model on the computed seismic response of flexible retaining structures Effets du coefficient d'amortissement et du model constitutif du sol sur la réponse sismique d'écrans de soutènement souples S. Aversa & R.M.S. Maiorano Università di Napoli "Parthenope", Naples, Italy C. Tamagnini Università di Perugia, Perugia, Italy

Influenza della modellazione di Rayleigh dello smorzamento viscoso nelle analisi di risposta sismica locale. Proc. XI Conf. "L'Ingegneria Sismica in Italia". Lanzo G., Pagliaroli A., D'Elia B. (2004).

MODELLING AND SIMULATION OF HEAVY TAMPING DYNAMIC RESPONSE OF THE GROUND Fritz Kopf 1, Ivan Paulmichl 2, Dietmar Adam 3(2010)

Proprietà viscolestiche e proprietà meccaniche dei polimeri, Elementi di viscoelasticità R Pantani.

Selection of Rayleigh Damping Coefficients for Seismic Response Analysis of Soil Layers Huai-Feng Wang, Meng-Lin Lou, Ru-Lin Zhang (2017)

Soil improvement using heavy tamping- a case history S.Kumar, V.K.puri (2001)

Appendix A) Siemens reports

==

Elenco di informazioni creato da: Z

Data : 07/02/2019 19:28:48

Parte di lavoro corrente : F:\assieme_iges.prt

Nome nodo : ez

==

Misurazione proprietà di massa

Visualizzati valori proprietà di massa

Volume = 1246.596926992 m^3

Area = 1797.034614848 m^2

Massa = 3116492.317481135 kg

Peso = 30562349.385226373 N

Raggio di inerzia = 12.622316044 m

Baricentro = 8.368002916, 8.899621613, 42.468174535 m

==

Proprietà di massa dettagliate

Analisi calcolata con una precisione di 0.990000000

Unità di misura informazioni kg - m

Densità = 2500.000000000

Volume = 1246.596926992

Area = 1797.034614848

Massa = 3116492.317481135

Primi momenti

Mx, My, Mz = 26078816.800225746, 27735602.384936336, 132351739.674481270

Centro di massa

BaricentroXc, BaricentroYc, BaricentroZc = 8.368002916, 8.899621613, 42.468174535

Momenti di inerzia (sistema di coordinate di lavoro)

Ix, Iy, Iz = 5940188173.312446600, 6265097775.974841100, 959372527.2
92508840

Momenti di inerzia (baricentro)

Ixc, Iyc, Izc = 72615026.430258214, 426133380.494168160, 494308545.833232700

Momenti di inerzia (sferico)

I = 496528476.378829540

Prodotto di inerzia (sistema di coordinate di lavoro)

Iyz, Ixz, Ixy = 1177746165.589488500, 1107173999.257709700, 232534289.
209149990

Prodotti di inerzia (baricentro)

Iyzc, Ixzc, Ixyc = -134237.315832097, -345744.269737673, 442687.576303222

Raggi giratori (sistema di coordinate di lavoro)

Rx, Ry, Rz = 43.658325157, 44.836416650, 17.545292330

Raggi di inerzia (baricentro)

Rxc, Ryc, Rzc = 4.827032463, 11.693371496, 12.594067861

Raggi di inerzia (sferico)

R = 12.622316044

Assi principali (direzione dei vettori relativa al sistema di coordinate di lavoro)

Xp(X), Xp(Y), Xp(Z) = 0.000817830, 0.001

963675, 0.999997738

Yp(X), Yp(Y), Yp(Z) = -0.001254149, 0.999997288, -0.001962649

Zp(X), Zp(Y), Zp(Z) = -0.999998879, -0.001252541, 0.000820291

Momenti principali

I1, I2, I3 = 494309092.193132160, 426133672.230294940, 72614188.334231988

===

Stima errore

Volume = 15.205212566

Area = 0.543249173

Massa = 38013.031415445

Raggio per centro di massa = 1.132374167

Momenti di inerzia (sistema di coordinate di lavoro) = 530537238.209506990, 526390317.065706250, 221023690.566638710

Momenti principali = 15341398.855725557, 118003297.187024830, 115246096.995264290

We want to simulate how the displacements and the velocities propagate in the media.

APPENDIX B) Thresholds laws

Table 1.B: UNI 9916:2014-Peak component particle velocity (p.c.p.v).

Valori di riferimento per la velocità di vibrazione (p.c.p.v.) al fine di valutare l'azione delle vibrazioni di breve durata sulle costruzioni

Classe	Tipo di edificio	Valori di riferimento per la velocità di vibrazione p.c.p.v in mm/s			
		Fondazioni			Piano alto
		Da 1 Hz fino a 10 Hz	Da 10 Hz fino a 50 Hz	Da 50 Hz fino a 100 Hz*)	Per tutte le frequenze
1	Costruzioni industriali, edifici industriali e costruzioni strutturalmente simili	20	Varia linearmente da 20 (*f*= 10 Hz) fino a 40 (*f*= 50 Hz)	Varia linearmente da 40 (*f*= 50 Hz) fino a 50 (*f*= 100 Hz)	40
2	Edifici residenziali e costruzioni simili	5	Varia linearmente da 5 (*f*= 10 Hz) fino a 15 (*f*= 50 Hz)	Varia linearmente da 15 (*f*= 50 Hz) fino a 20 (*f*= 100 Hz)	15
3	Costruzioni che non ricadono nelle classi 1 e 2 e che sono degne di essere tutelate (per esempio monumenti storici)	3	Varia linearmente da 3 (*f*= 10 Hz) fino a 8 (*f*= 50 Hz)	Varia linearmente da 8 (*f*= 50 Hz) fino a 10 (*f*= 100 Hz)	8

*) Per frequenze oltre 100 Hz possono essere usati i valori di riferimento per 100 Hz.

Table 2.B Peak particle velocities

Valori di riferimento della velocità di vibrazione (p.p.v.) al fine di valutare l'azione delle vibrazioni transitorie sulle costruzioni

Classe	Tipo di costruzione	Esposizione	Valori di riferimento per la velocità di vibrazione p.p.v. in mm/s		
			Posizioni di misura *)		
			Da 8 Hz fino a 30 Hz *)	Da 30 Hz fino a 60 Hz	Da 60 Hz fino a 150 Hz
A	Costruzioni molto poco sensibili (per esempio ponti, gallerie, fondazioni di macchine)	Occasionale Frequente Permanente	Fino a tre volte i valori corrispon-denti alla classe C	Fino a tre volte i valori corrispon-denti alla classe C	Fino a tre volte i valori corrispon-denti alla classe C
B	Costruzioni poco sensibili (per esempio edifici industriali in cemento armato o metallici) costruiti a regola d'arte e con manutenzione adeguata	Occasionale Frequente Permanente	Fino a due volte i valori corrispon-denti alla classe C	Fino a due volte i valori corrispon-denti alla classe C	Fino a due volte i valori corrispon-denti alla classe C
C	Costruzioni normalmente sensibili (per esempio edifici d'abitazione in muratura di cemento, cemento armato o mattoni, edifici ammini-strativi, scuole, ospedali, chiese in pietra naturale o mattoni intonacati) costruiti a regola d'arte e con manutenzione adeguata	Occasionale Frequente Permanente	15 6 3	20 8 4	30 12 6
D	Costruzioni particolarmente sensibili (per esempio monumenti storici e soggetti a tutela) case con soffitti in gesso, edifici della classe C nuovi o ristrutturati di recente	Occasionale Frequente Permanente	Valori compresi tra quelli previsti per la classe C e la loro metà	Valori compresi tra quelli previsti per la classe C e la loro metà	Valori compresi tra quelli previsti per la classe C e la loro metà

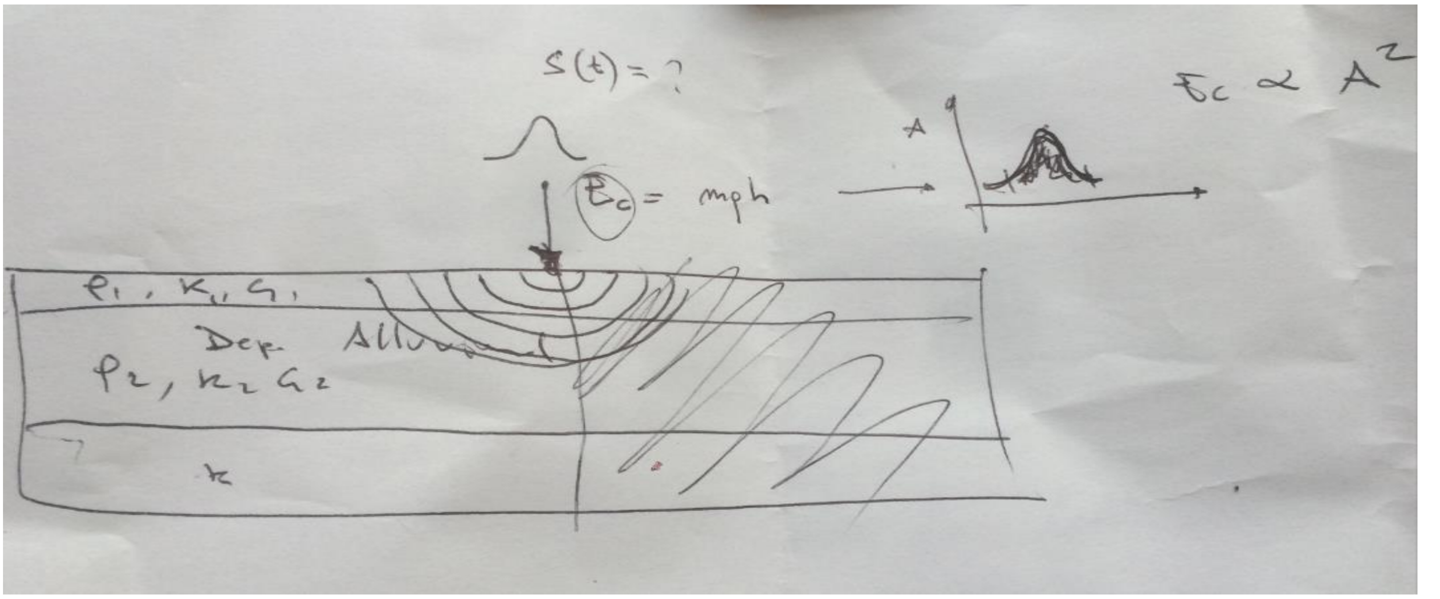
*) Le posizioni di misura devono essere scelte sugli elementi rigidi della struttura portante o dove sono attesi i maggiori effetti delle vibrazioni.

APPENDIX C) Matlab's codes

Programm 1)

2D Structural modelling

our model




```
%create a model in strucutral mechanics analysis with transient study of
%the behaviour. the geometry analysed is that in the figure above. It's
%thought as a multucuboid strata layers.
model = createpde('structural', 'transient-solid');
geo = multicuboid(15,200,[10 10 10],'ZOffset',[0 10 20]);
model.Geometry = geo;
pdegplot(model,'FaceLabels','on','FaceAlpha',0.5)
%return
%Each stratum has its own physical properties, young Modulus, Density and
%Poisson ratio. The layers are identifying as cells.
structuralProperties(model,'cell',1,'YoungsModulus',450E6,'MassDensity',1800,'PoissonsRatio',0.375);
structuralProperties(model,'cell',2,'YoungsModulus',389E6,'MassDensity',1800,'PoissonsRatio',0.15);
structuralProperties(model,'cell',3,'YoungsModulus',705E6,'MassDensity',1900,'PoissonsRatio',0.16);
%The boundary on x ae considered fixed in the ground surrounding.
% structuralBC(model,'Constraint','fixed','Face',16);
% structuralBC(model,'Constraint','fixed','Face',14);
```

```
% structuralBC(model,'Constraint','fixed','Face',11);

% structuralBC(model,'Constraint','fixed','Face',9);

% structuralBC(model,'Constraint','fixed','Face',4);

% structuralBC(model,'Constraint','fixed','Face',6);

%boundary condition on face 1=>face one is considered fixed.

structuralBC(model,'Constraint','fixed','Face',1);

%generete the mesh grid

generateMesh(model,'Hmax',5);

pdemesh(model,'FaceAlpha',0.5)

% Define the load, we define it as a pressure acting on face 16.

structuralBoundaryLoad(model,'Face',16,'Pressure',485200,'StartTime',0,'EndTime',0.010)

% Initial conditions

structuralIC(model,'Displacement',[0;0;0],'Velocity',[0;0;0]);

% Time frames

tlist = linspace(0,0.15,5);

result = solve(model,tlist);

pdeplot3D(model,'ColorMapData',result.Displacement.uz(:,2))
```

Programm 2)

```
%create a model in strucutral mechanics analysis with transient study of
%the behaviour. the geometry analysed is that in the figure above. It's
%thought as a multucuboid strata layers.
model = createpde('structural', 'transient-solid');
geo = multicuboid(45,20,100);
model.Geometry = geo;
pdegplot(model,'FaceLabels','on','FaceAlpha',0.5)
%return

%Each stratum has its own physical properties, young Modulus, Density and
%Poisson ratio. The layers are identifying as cells.
structuralProperties(model,'cell',1,'YoungsModulus',450E6,'MassDensity',1800,'PoissonsRatio',0.375);
%The boundary on x ae considered fixed in the ground surrounding.
structuralBC(model,'Constraint','fixed','Face',1);

%generete the mesh grid

generateMesh(model,'Hmax',5);
pdemesh(model,'FaceAlpha',0.5)

% Define the load, we define it as a pressure acting on face 16.
```

```
structuralBoundaryLoad(model,'Face',2,'Pressure',1150000,'StartTime',0,'EndTime',0.
050)

% Initial conditions


structuralIC(model,'Displacement',[0;0;0],'Velocity',[0;0;0]);

% Time frames



tlist = linspace(0,0.15,5);
result = solve(model,tlist);
pdeplot3D(model,'ColorMapData',result.Displacement.uz(:,2))
```

Program 3)

```
%create a model in strucutral mechanics analysis with transient study of
%the behaviour. the geometry analysed is that in the figure above. It's
%thought as a multucuboid strata layers.
model = createpde('structural', 'transient-solid');
geo = multicuboid(50,20,[100 3],'ZOffset',[0 100]);
% geo = multicuboid(45,20,100);
model.Geometry = geo;
pdegplot(model,'FaceLabels','on','FaceAlpha',0.5)
%return

%Each strata has its own physical properties, young Modulus, Density and
%Poisson ratio. The layers are identifies as cells.
structuralProperties(model,'cell',1,'YoungsModulus',450E6,'MassDensity',1800,'Poiss
onsRatio',0.375);
structuralProperties(model,'cell',2,'YoungsModulus',40E6,'MassDensity',1500,'Poisso
nsRatio',0.3);
%The boundary on x ae considered fixed in the ground surrounding.
structuralBC(model,'Constraint','fixed','Face',1);

%generete the mesh grid
generateMesh(model,'Hmax',5);
pdemesh(model,'FaceAlpha',0.5)

% Define the load, we define it as a pressure acting on face 16.


structuralBoundaryLoad(model,'Face',7,'Pressure',1150000,'StartTime',0,'EndTime',0.
20)
% Initial conditions
structuralIC(model,'Displacement',[0;0;0],'Velocity',[0;0;0]);

% Time frames

tlist = linspace(0,0.15,5);
result = solve(model,tlist);
pdeplot3D(model,'ColorMapData',result.Displacement.uz(:,2))
```

Program 4)

```
clear all
model = createpde('structural','static-planestress');
%mettere la geometria
r1 = [3 4 0 30 30 0 0 0 50 50];
r2 = [3 4 14 16 16 14 46 46 48 48];
r3 = [3 4 -30 60 60 -30 0 0 50 50];
gdm = [r1; r2; r3]';
ns = char('r1','r2','r3');
g = decsg(gdm,'(R3+R1)-R2',['R1'; 'R2'; 'R3']');
geometryFromEdges(model,g);
figure
pdegplot(model,'FaceLabels','on');
axis equal
title 'Geometry with Edge Labels';
%  return
structuralProperties(model,'YoungsModulus',450E6,'PoissonsRatio',0.375,'MassDensity
',1800);
%
structuralProperties(model,'Cell',2,'YoungsModulus',25E9,'PoissonsRatio',0.3,'MassD
ensity',2500);
structuralBC(model,'Edge',9,'Constraint','roller');
structuralBC(model,'Edge',5,'Constraint','roller');
structuralBC(model,'Edge',10,'Constraint','roller');
structuralBC(model,'Edge',11,'Constraint','roller');
structuralBC(model,'Edge',4,'Constraint','roller');
structuralBoundaryLoad(model,'Edge',6,'Pressure',-100);
structuralBoundaryLoad(model,'Edge',8,'Pressure',-100);
structuralBoundaryLoad(model,'Edge',7,'Pressure',-100);
structuralBoundaryLoad(model,'Edge',7,'Pressure',-400000);
generateMesh(model,'Hmax', 0.2);
figure
pdemesh(model)
R = solve(model);
figure
pdeplot(model,'XYData',R.VonMisesStress,'ColorMap','jet')
axis equal
xlabel('asse x')
ylabel('asse y')
title('stress su x')
figure
pdeplot(model,'XYData',R.Stress.syy,'ColorMap','jet')
axis equal
xlabel('asse x')
ylabel('asse y')
title 'Normal Stress Along y-Direction';
 save tubazione_result.mat R -mat; save tubazione_model.mat model -mat
```