Analysis of Offshore Protective Berms using FEM and MPM

MSc Thesis Report
Geo-Engineering

By
Shayan Kalanaki
January 2018
Analysis of Offshore Protective Berms using FEM and MPM

Protective Berms for Offshore Structures against Ship Collisions

by

Shayan Kalanaki

January 2018

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Civil Engineering, Geo-Engineering specialisation at the Delft University of Technology

*An electronic version of this thesis is available at: http://repository.tudelft.nl/*
Author:

Shayan Kalanaki,
St. NO.: 4086643

Thesis Committee Members:

Dr. Ir. R. B. J. Brinkgreve, Department of Geo-Engineering
Delft University of Technology

Dr. P. J. Vardon, Department of Geo-Engineering
Delft University of Technology

Prof. Dr. Ir. M. L. Kaminski, Department of Ship Hydromechanics & Structures
Delft University of Technology

Ir. A. W. Van Houwelingen, Department of Maritime & Aviation
Royal HaskoningDHV
PREFACE

This report presents my research on analysis of Offshore Protective Berms using Finite Element Method (FEM) and Material Point Method (MPM) and has been submitted as the last part of the curriculum of the Master of Science in Geo-Engineering at Delft University of Technology (TU Delft). This research has been completed at TU Delft, Royal HaskoningDHV Amersfoort Maritime & Aviation department and Plaxis Delft office.

Due to my interest in Offshore Geotechnical Engineering and the usage of the recently developed MPM, which is considered an important numerical calculation tool for the future, I chose this topic and accepted the possible challenges.

I would like to express my gratitude to Ir. A. W. Van Houwelingen for introducing me to the topic, providing me the opportunity to complete this research at Royal HaskoningDHV and for his guidance. Furthermore, I would like to thank my university supervisors comprising of Dr. Ir. R. B. J. Brinkgreve and Dr. P. J. Vardon from Geo-Engineering section for their guidance and suggestions during my research and also Prof. Dr. Ir. M. L. Kaminski from Ship Hydromechanics & Structures section. Also, I would like to thank other persons who were involved in my thesis, such as Ir. R. W. Bos from TU Delft and especially Ir. A. Sarathchandran for his substantial amount of guidance during my time at Plaxis office and while using the MPM software for modelling.

Last but not least, I wish to express my special appreciation to my parents and brother who always supported me throughout my life and studies.

Shayan Kalanaki
Delft, January 2018
ABSTRACT

By increase in knowledge and complexity of the projects, worldwide, more attention towards safety provision in projects is being paid. With growth in waterway traffic the need of safety provision for structures crossing navigable waters increases. Especially bridge piers and immersed tunnels on the see floor or at tunnel approaches are at bigger risk of being hit by ships which have lost their navigational course due to any reasons. Therefore availability of collision protective systems is essential. A number of different methods are being used worldwide as protective systems while for the full frontal collision, protective berms are the most convenient systems. This research investigates the possibility of calculations for these berms using numerical modelling tools such as Finite Element Method (FEM) and Material Point Method (MPM). FEM is a well-known and proven tool in engineering world while MPM is a newly developed tool that is considered as an update of FEM. In MPM the material state and stresses are stored in material points that can freely move though the mesh. This tool is highly useful for problems with high expected deformations or dynamic calculations where FEM is not considered as an ideal tool and could fail. Furthermore, also, a simplified analytical method has been presented in this report.

At present MPM is being developed around the world. Plaxis, which is a well-known software developer especially for Geo-Engineering applications, is currently developing software that provides MPM modelling. For this research, the Plaxis 2D software has been used for MPM under confidentiality terms. As mentioned, the software is in its development stage therefore some issues where expected. The FEM models are also created using the internationally used Plaxis 2D FEM version.

The reliability of the MPM and also FEM models in dynamic calculations was analyzed in this research allowing for a better understanding for the final model which represents the ship-berm collision problem. Two models (static and dynamic) were created in both FEM and MPM which then verified and compared. Therefore it was concluded that MPM can be considered as a reliable tool for dynamic calculations since the obtained results were similar or comparable with FEM and analytical solutions.

The final models in both FEM and MPM however, did not provide the required results that could help to understand the ship-berm collision mechanism and/or the stress situations within the soil (only partial results were obtained). The FEM model showed very unrealistic and inaccurate results. The constant predefined contact area between the ship and berm is considered to be the main issue in this model; on the other hand, due to encountered issues with MPM, the MPM simulation could not be fully performed in terms of minimum required time steps. Nevertheless the partial simulations with a limited simulation time, were found to be realistic and in accordance with expectation, suggesting the capability of MPM for dynamic calculation after the necessary developments.

As conclusion, this report does not recommend the usage of FEM for such a modelling and considers MPM a good possible tool to be used after the necessary developments in the near future. Meanwhile the presented simplified analytical method could provide a rough estimation for required berm length. In addition a FEM model has also been suggested in this report allowing for the user to have an insight on the stress propagation within the soil.
6.2. Recommendation for future research ...................................................... 53

Bibliography ........................................................................................................... 55

Appendices ................................................................................................................ 57
  Appendix A: FEM horizontal dynamic initial checks ............................................ 57
  Appendix B: Dynamic FEM sensitivity analysis .................................................. 62
  Appendix C: Dynamic FEM vs MPM ................................................................. 65
  Appendix D: Steps 2 and 3 of the calculation sample ........................................ 68
## LIST OF FIGURES AND TABLES

Figure 1: The Sunshine Skyway Bridge deck collapse due to ship collision, Florida, USA
[Source: https://en.wikipedia.org/wiki/MV_Summit_Venture] ............................................. 1
Figure 2: Ship-bridge collisions incidents [6] ........................................................................ 2
Figure 3: Deflection and rotation of dolphin [1] ..................................................................... 7
Figure 4: Island protective system [1] .................................................................................. 8
Figure 5: Failure mechanism ................................................................................................. 11
Figure 6: Forces on ship bow [15] ........................................................................................ 11
Figure 7: Ship's 6-Degrees of freedom [19] ......................................................................... 13
Figure 8: MPM calculation phases [3] .................................................................................. 19
Figure 9: Stress oscillation for 4 noded elements (left) and 8 noded elements (right) [7] ...... 20
Figure 10: Plastic model ........................................................................................................ 23
Figure 11: Prandtl's schematization [22] .............................................................................. 23
Figure 12: Stresses in zone III [22] ...................................................................................... 24
Figure 13: FEM Elasto-Plastic model result .......................................................................... 24
Figure 14: FEM and MPM Elasto-Plastic model result ............................................................ 25
Figure 15: Linear elastic Plaxis model ................................................................................... 26
Figure 16: Displacement and velocity vs time for linear elastic model .................................. 27
Figure 17: Vertical vs Horizontal dynamic model ................................................................. 29
Figure 18: MPM and FEM soil displacement ......................................................................... 30
Figure 19: MPM and FEM block velocity ............................................................................. 30
Figure 20: MPM dynamic model .......................................................................................... 31
Figure 21: FEM ship-berm model ......................................................................................... 32
Figure 22: FEM ship-berm model result (true scale) ............................................................... 35
Figure 23: FEM final model ship displacement .................................................................... 36
Figure 24: FEM final model ship velocity ............................................................................ 36
Figure 25: FEM soil failure at approximately 0.2 s ............................................................... 37
Figure 26: FEM final situation after 5 seconds (true scale) ..................................................... 37
Figure 27: MPM ship-berm displacement .......................................................................... 38
Figure 28: MPM final model ship velocity .......................................................................... 38
Figure 29: MPM shear stress before failure ....................................................................... 39
Figure 30: MPM failure plane ............................................................................................. 39
Figure 31: MPM second failure plane ............................................................................... 40
Figure 32: Shear stress vs time ............................................................................................ 40
Figure 33: MPM ship-berm horizontal stress ..................................................................... 41
Figure 34: Deformation after approximation 0.5 seconds in both FEM and MPM ............. 42
Figure 35: Ship anchor berm model .................................................................................... 43
Figure 36: Elasto-Plastic vs Elastic ....................................................................................... 44
Figure 37: Suggested FEM model ....................................................................................... 45
Figure 38: Sample calculation Plaxis model ....................................................................... 48
Figure 39: Suggested model displacement comparison ....................................................... 49
Figure 40: Suggested model stress comparison .................................................................. 49
Figure 41: Vertical displacement check ............................................................................. 57
Figure 42: Vertical stress check .......................................................................................... 58
Figure 43: Horizontal displacement check ........................................................................ 59
Figure 44: Horizontal velocity check .................................................................................. 59
Figure 45: Horizontal stress check ..................................................................................... 60
Figure 46: Horizontal displacement after 0.1 second ........................................................... 60
Figure 47: Displacement vs time for stiffness changes ....................................................... 62
Figure 48: Velocity vs time for stiffness changes .......................................................... 62
Figure 49: Displacement vs time for mass changes .......................................................... 63
Figure 50: Velocity vs time for mass changes ................................................................. 63
Figure 51: Displacement vs time for velocity changes ....................................................... 64
Figure 52: Velocity vs time for velocity changes ......................................................... 64
Figure 53: FEM and MPM soil displacement vs time, dataset 1 ........................................ 65
Figure 54: FEM and MPM block velocity vs time, dataset 1 ........................................... 65
Figure 55: FEM and MPM soil displacement vs time, dataset 2 ....................................... 66
Figure 56: FEM and MPM block velocity vs time, dataset 2 ........................................... 66
Figure 57: FEM and MPM soil displacement vs time, dataset 2 ....................................... 67
Figure 58: FEM and MPM block velocity vs time, dataset 3 ........................................... 67

Table 1: Advantages and disadvantages of available protection methods ......................... 9
## Abbreviations and Symbols

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AASHTO</td>
<td>American Association of State Highway and Transportation Officials</td>
</tr>
<tr>
<td>DDMP</td>
<td>Dual Domain Material Point</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Method</td>
</tr>
<tr>
<td>LE</td>
<td>Linear Elastic</td>
</tr>
<tr>
<td>MC</td>
<td>Mohr Coulomb</td>
</tr>
<tr>
<td>MPM</td>
<td>Material Point Method</td>
</tr>
<tr>
<td>RAT</td>
<td>Rambøll Arup TEC</td>
</tr>
<tr>
<td>α</td>
<td>Slope angle [°]</td>
</tr>
<tr>
<td>ε</td>
<td>Strain [-]</td>
</tr>
<tr>
<td>ε&lt;sub&gt;v&lt;/sub&gt;</td>
<td>Total vertical strain [-]</td>
</tr>
<tr>
<td>ϕ</td>
<td>Internal friction angle [°]</td>
</tr>
<tr>
<td>ϕ&lt;sub&gt;m&lt;/sub&gt;</td>
<td>Modelled soil internal friction angle [°]</td>
</tr>
<tr>
<td>ϕ&lt;sub&gt;soil&lt;/sub&gt;</td>
<td>To be modelled soil internal friction angle [°]</td>
</tr>
<tr>
<td>γ</td>
<td>Unit weight [kN/m&lt;sup&gt;3&lt;/sup&gt;]</td>
</tr>
<tr>
<td>γ&lt;sub&gt;m&lt;/sub&gt;</td>
<td>Soil unit weight in the model [kN/m&lt;sup&gt;3&lt;/sup&gt;]</td>
</tr>
<tr>
<td>γ&lt;sub&gt;water&lt;/sub&gt;</td>
<td>Unit weight of water [kN/m&lt;sup&gt;3&lt;/sup&gt;]</td>
</tr>
<tr>
<td>μ</td>
<td>Friction coefficient [-]</td>
</tr>
<tr>
<td>π</td>
<td>Pi [-]</td>
</tr>
<tr>
<td>σ</td>
<td>Stress [kN/m&lt;sup&gt;2&lt;/sup&gt;]</td>
</tr>
<tr>
<td>σ&lt;sub&gt;m&lt;/sub&gt;</td>
<td>Model stress [kN/m&lt;sup&gt;2&lt;/sup&gt;]</td>
</tr>
<tr>
<td>σ&lt;sub&gt;T&lt;/sub&gt;</td>
<td>Total stress [kN/m&lt;sup&gt;2&lt;/sup&gt;]</td>
</tr>
<tr>
<td>σ&lt;sub&gt;xy&lt;/sub&gt;</td>
<td>Shear stress [kN/m&lt;sup&gt;2&lt;/sup&gt;]</td>
</tr>
<tr>
<td>A</td>
<td>Area [m&lt;sup&gt;2&lt;/sup&gt;]</td>
</tr>
<tr>
<td>a</td>
<td>Width in created models [m]</td>
</tr>
<tr>
<td>d</td>
<td>Ship draft [m]</td>
</tr>
<tr>
<td>E</td>
<td>Stiffness [kN/m&lt;sup&gt;2&lt;/sup&gt;]</td>
</tr>
<tr>
<td>E&lt;sub&gt;f&lt;/sub&gt;</td>
<td>Frictional energy [kJ]</td>
</tr>
<tr>
<td>E&lt;sub&gt;kin&lt;/sub&gt;</td>
<td>Kinetic energy [kJ]</td>
</tr>
</tbody>
</table>
\( E_{\text{kin,m}} \)  \hspace{1cm} Kinetic energy of the model \hspace{1cm} [kJ]

\( \Delta E_{\text{kin}} \)  \hspace{1cm} Changes in kinetic energy \hspace{1cm} [kJ]

\( E_{\text{pot}} \)  \hspace{1cm} Potential energy \hspace{1cm} [kJ]

\( E_T \)  \hspace{1cm} Total energy \hspace{1cm} [kJ]

\( F \)  \hspace{1cm} Force \hspace{1cm} [kN]

\( F_H \)  \hspace{1cm} Horizontal force \hspace{1cm} [kN]

\( F_m \)  \hspace{1cm} Force in the model \hspace{1cm} [kN]

\( F_V \)  \hspace{1cm} Vertical force \hspace{1cm} [kN]

\( g \)  \hspace{1cm} Gravitational acceleration \hspace{1cm} [m/s\(^2\)]

\( H \)  \hspace{1cm} Height \hspace{1cm} [m]

\( Heave^* \)  \hspace{1cm} Non-definitive ship heave \hspace{1cm} [m]

\( Heave \)  \hspace{1cm} Definitive ship heave \hspace{1cm} [m]

\( H_{\text{bow}} \)  \hspace{1cm} Height of ship bow \hspace{1cm} [m]

\( H_{\text{keel}} \)  \hspace{1cm} Height of ship keel \hspace{1cm} [m]

\( H_{\text{keel,max}} \)  \hspace{1cm} Maximum height of ship keel \hspace{1cm} [m]

\( H_{\text{crest}} \)  \hspace{1cm} Height of ship crest \hspace{1cm} [m]

\( H_{\text{stern}} \)  \hspace{1cm} Height of ship stern \hspace{1cm} [m]

\( i \)  \hspace{1cm} Calculations step \hspace{1cm} [-]

\( L \)  \hspace{1cm} Length \hspace{1cm} [m]

\( \Delta L \)  \hspace{1cm} Compression \hspace{1cm} [m]

\( L_m \)  \hspace{1cm} Length of the model \hspace{1cm} [m]

\( L_{\text{ship}} \)  \hspace{1cm} Ship length \hspace{1cm} [m]

\( m \)  \hspace{1cm} Mass \hspace{1cm} [kg]

\( m_{\text{add}} \)  \hspace{1cm} Added mass \hspace{1cm} [kg]

\( n \)  \hspace{1cm} Scale factor \hspace{1cm} [-]

\( Pitch \)  \hspace{1cm} Ship pitch \hspace{1cm} [°]

\( q \)  \hspace{1cm} Line load \hspace{1cm} [kN/m]

\( q_{\text{max}} \)  \hspace{1cm} Maximum resisting line load \hspace{1cm} [kN/m]

\( R_{\text{inter}} \)  \hspace{1cm} Interface strength \hspace{1cm} [-]
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S_u$</td>
<td>Undrained shear strength</td>
<td>[kN/m$^2$]</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
<td>[s]</td>
</tr>
<tr>
<td>$u$</td>
<td>Displacement</td>
<td>[m]</td>
</tr>
<tr>
<td>$V$</td>
<td>Velocity</td>
<td>[m/s]</td>
</tr>
<tr>
<td>$V_m$</td>
<td>Block velocity</td>
<td>[m/s]</td>
</tr>
<tr>
<td>$V_{ship}$</td>
<td>Ship velocity</td>
<td>[m/s]</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Poisson ratio</td>
<td>[-]</td>
</tr>
<tr>
<td>$W$</td>
<td>Work</td>
<td>[kN.m]</td>
</tr>
<tr>
<td>$W_m$</td>
<td>Work done in the model</td>
<td>[kN.m]</td>
</tr>
<tr>
<td>$\Delta W$</td>
<td>Changes in work</td>
<td>[kN.m]</td>
</tr>
<tr>
<td>$w$</td>
<td>Width</td>
<td>[m]</td>
</tr>
<tr>
<td>$w_{ship}$</td>
<td>Ship width</td>
<td>[m]</td>
</tr>
<tr>
<td>$x$</td>
<td>Ship horizontal movement</td>
<td>[m]</td>
</tr>
<tr>
<td>$x_m$</td>
<td>The distance that the force is being applied</td>
<td>[m]</td>
</tr>
</tbody>
</table>
**EQUATIONS**

\[ E_{\text{kin}} = 0.5 \cdot (m + m_{\text{add}}) \cdot V_{\text{ship}}^2 \quad [kJ] \]  
\[ \text{Heave}(i) = \frac{H_{\text{bow}}(i) + H_{\text{stern}}(i)}{2} \quad [m] \]  
\[ \text{Pitch}(i) = \arctan \left( \frac{H_{\text{bow}}(i) + H_{\text{stern}}(i)}{L_{\text{ship}}} \right) \quad [^\circ] \]  
\[ H_{\text{bow}}(i) = \frac{x(i)}{\cot(\alpha)} \quad [m] \]  
\[ H_{\text{stern}}(i) = \frac{H_{\text{bow}}(i)}{2} \quad [m] \quad \text{For } x(i) \leq \frac{2}{3} L_{\text{ship}} \]  
\[ H_{\text{stern}}(i) = H_{\text{stern}}(i - 1) + \left( x(i) - \frac{2}{3} L_{\text{ship}} \right) \cdot \left( \frac{1}{\cot(\alpha)} \right) \quad [m] \quad \text{For } x(i) > \frac{2}{3} L_{\text{ship}} \]  
\[ \text{Heave}(i) = \text{Heave}(i - 1) \quad [m] \quad \text{if } H_{\text{Keel}} = H_{\text{crest}} \]  
\[ \text{Heave}(i) = \text{Heave}(i - 1) \quad [m] \quad \text{if } H_{\text{Keel}} \neq H_{\text{crest}} \]  
\[ H_{\text{keel}} = \min \left( (H_{\text{keel,max}} + \text{Heave}), H_{\text{crest}} \right) \quad [m] \]  
\[ F_V(i) = \text{Heave}(i) \cdot w_{\text{ship}} \cdot L_{\text{ship}} \cdot \gamma_{\text{water}} \quad [kN] \]  
\[ F_H(i) = F_V(i) \cdot \mu \quad [kN] \]  
\[ E_{\text{pot}}(i) = m \cdot g \cdot \text{Heave}(i) + E_{\text{pot}}(i - 1) \quad [kJ] \]  
\[ E_f(i) = 0.5 \cdot (F_H(i) + F_H(i - 1)) \cdot (x(i) - x(i - 1)) + E_f(i - 1) \quad [kJ] \]  
\[ E_T = \sum E_{\text{pot}}(i) + \sum E_f(i) \quad [kJ] \]  
\[ \mu = \frac{2}{3} \cdot \tan(\varphi) \quad [-] \]  
\[ q_{\text{max}} = (2 + \pi) \cdot S_u \quad [\frac{kN}{m^2}] \]  
\[ \sigma = \frac{F}{A} \quad [\frac{kN}{m^2}] \]
\[
\varepsilon = \frac{\sigma}{E} \quad [-]
\]

\[
W = (-F + mg) \Delta L \quad [kN.m]
\]

\[
\Delta W = \Delta E_{kin} = 0 - \frac{1}{2} mV^2 \quad [kN.m or kJ]
\]

\[
F = m \left( g + \frac{V^2}{2\Delta L} \right) \quad [kN]
\]

\[
\sigma_T = \varepsilon_e E \quad \left[ \frac{kN}{m^2} \right]
\]

\[
F = A\sigma_T \quad [kN]
\]

\[
W = F \cdot x \quad [kN.m]
\]

\[
L_m = \frac{L}{n} \quad [m]
\]

\[
V_m = \frac{V}{s}
\]

\[
E_{kin,m} = W_m = F_m \cdot x_m = \sigma_m \cdot a \cdot x_m \quad [kJ]
\]

\[
m_m = \frac{2 \cdot E_{kin,m}}{V_m^2} \quad [kg]
\]

\[
\gamma_m = \frac{m_m \cdot g}{a^2} \quad \left[ \frac{kN}{m^3} \right]
\]

\[
E_m = \frac{\sigma_m}{\varepsilon} = \frac{\sigma_m}{x_m} = \frac{\sigma_m \cdot L_m}{x_m} \quad \left[ \frac{kN}{m^2} \right]
\]

\[
\tan(\varphi_{soil}) = R_{inter} \cdot \tan(\varphi_{model}) \quad [-]
\]
CHAPTER 1: GENERAL INTRODUCTION

1.1. INTRODUCTION

In modern engineering world, more attentions are being paid towards the safety provision while the knowledge is increasing and as the projects are becoming more complex. The bridges and underwater tunnels, crossing navigable waters are not distinct from this fact. One of the main concerns for these projects is the risk of ship collisions and/or grounding ships impact on the structures.

In many busy waterways, where also large ships navigate, waterway crossing structures and structures placed adjacent to the waterways are in risk of ship collisions. The possibility of a ship losing its main navigational course always exists. By growth in navigation traffic, for bridges crossing the waterways, ship and barge collisions are considered as serious and growing threats. Therefore these structures should be protected in a way.

Worldwide, in a period of 42 years (from 1960 to 2002), 31 major incidents involving life casualties, with 342 deaths, have been recorded due to ship colliding with bridges [1]. The number of incidents will be increased dramatically if the incidents with no life losses are also taken into account. Figure 1 shows the tragic incident in Tampa Bay, Florida, where the bridge deck of the Sunshine Skyway Bridge was hit by an empty bulk carrier causing the deck to collapse resulting in 35 lost lives.

![Figure 1: The Sunshine Skyway Bridge deck collapse due to ship collision, Florida, USA](https://en.wikipedia.org/wiki/MV_Summit_Venture)

[Source: HTTPS://EN.WIKIPEDIA.ORG/WIKI/MV_SUMMIT_VENTURE]
With growth in ship navigation, the incident trends are consequently increasing yearly. This is evident from the increases in number of serious incidents shown by blue line in the presented graph (Figure 2 [6]). This graph represents data till the year 1998; however the trend is still increasing up to year 2017.

![Graph of ship-bridge collision incidents](image)

**FIGURE 2: SHIP-BRIDGE COLLISIONS INCIDENTS [6]**

The incidents in 2007 and 2013 at Bay Bridge in United States are more recent examples of ship-bridge collisions. The incident in 2007 led to no human casualty however the spilling oil was responsible for the death of thousands of birds; and the 2013 incident resulted in damages to ship and bridge pier protective systems.

A number of factors such as poor visibility, poorly sited bridge, insufficient attention while navigating and bad weather conditions could be the reasons for ship-bridge collisions. However, aside from these, the human error is considered as the most common cause.

Therefore sufficient attention has to be paid regarding the safety against ship collisions. Based on the waterway information and the ships navigating under the bridge or on top of a tunnel, a number of minimum requirements will be determined for each project.

For bridge decks a possible mitigation against ship mast impact is to build the bridge higher than the maximum height of the passing ships based on the data. Although this issue and solutions are irrelevant to the topic of this report, they are still important factors in the designing phase. Nevertheless increasing the deck height may not be always possible due to the higher costs and other limitations. In some cases the bridge decks are designed in a way to withstand the impact forces of ships without any serious damage intake.

Bridge piers and immersed tunnels are different stories since in case of a collision relatively high forces are applied to the structure as the whole ship’s mass is involved. If no alternative ship navigational routes are possible or no preventive measures are designed, the pier itself needs to be strong enough to stay intact if being hit by ship or in case of an immersed tunnel, the tunnel outer layer and the joints should be strong enough to overcome the applied forces.

In most projects, the engineers do not take the risk of allowing the ships to reach the piers or main tunnel structures. Therefore availability of protective systems in waterways is very common. A number of used methods, protecting the bridge piers and immersed tunnels against ships will be presented in chapter 2. The main focus of this report is on the method
which uses a berm, made by fill materials (mainly sand) to stop the ship before reaching the main structure or reduce the impact forces as much as possible. This method will be explained in detail.

These protective berms are not only used for bridge piers they are also considered as a physical protection for underwater tunnels. Tunnels lying on the seabed (e.g. immersed tunnels) are in danger of being hit by ships in case of a shallow water depth. Another danger regarding tunnels is the ship anchors. Anchors may be dropped on top of the tunnel or even being pulled afterwards by still moving ships. The tunnel approaches should also be protected against similar dangers. The protective measures at tunnel approaches are in a way similar to the bridge pier being protected by berms. Therefore the topic of this research covers both water crossing bridges and tunnels.

One of the main concerns here is the modelling/calculation of the forces being applied by the ship in case of a collision. It is also essential to know the stopping distance of the ship on the engineered berm including the horizontal and friction forces. By knowing the minimum required stoppage distance the protective berm can be constructed as small as possible which results in a more economical berm and is certainly of an importance where the space and availability of the material is limited.

The horizontal forces generated by ship collision are mainly “checking” conditions for structural loading (of tunnel or bridge pier) and currently estimated assuming a regular spreading of force and a uniform transfer of loads through the berm, ignoring 3 dimensional effects and variations in stiffness between berm and structure.

Although currently some tools such as national or international design codes are available for calculations and designs, these tools do not provide all the necessary data and/or results are considered to be conservative for the engineering practices. Therefore due to importance of these protective islands and not existence of a suitable calculation tool, this research analysis the calculation possibilities using mainly numerical methods.

In chapter 2 of this report, relevant information regarding the research topic will be given with in detailed explanations of collision berms in particular followed by analytical calculation method. Further the two used numerical methods, namely Finite Element Method (FEM) and Material Point Method (MPM), will be explained.

### 1.2. **Research Objectives and Methodology**

The main focus of this research is on the so called collision berms, the protective fill material surrounding the structures that are in risk of ship collisions. As briefly mentioned in previous section, it is important to know the required berm length to bring the colliding ship into a full stop. The horizontal forces and stresses that will travel and propagate through the berm and finally reaching the protected structure are other important aspects.

Since the required parameters and information are difficult to calculated using analytical solutions, numerical modelling is suggested to be the best approach. Therefore the main goal of this research is **calculation of ship-berm collision using numerical modelling**. This is considered to be the ideal outcome of this research however due to complexity of the problem and its difficulty to model, the main objective may alter to **analyzing the possibility of collision berm calculation using numerical modelling** namely FEM and MPM. All the numerical calculations whether using FEM or MPM in this research are
performed using Plaxis 2D software. The MPM software is still an unfinished product and is still under development.

Aside from the main goal a couple of side objectives have also been set, namely:

- Analyzing the available analytical methods
- Performing checks and verification on numerical models
- Comparing the analytical and numerical methods

The analytical calculation method uses a highly simplified approach. These simplifications are necessary to allow engineers to have a rough estimation for the required berm length. However this method does not provide any information about the stresses and deformations within the berm. Therefore the importance of the numerical modelling in this research becomes more significant.

In order to be able to create a reliable final model, the two basis of the model, which are dynamic calculations and Elasto-plastic deformations, should be checked and verified. Therefore firstly two simplified models in FEM will be made and after verifications of the results, they will be checked with the results from MPM. This approach allows for a comparison and reliability check for MPM that is still under development.

As mentioned above, the numerical modelling starts with a simple model that follows the Mohr Coulomb criteria which includes both plastic and elastic deformations. This will be done based on static calculations that provides, already proven to be reliable, results in FEM with the goal of verification of MPM results.

Further a dynamic calculation using a linear elastic material will be performed. Due to lack of experience with dynamic FEM calculation, a simple dynamic model that can be verified by analytical solutions is a good starting point. After the verification of this model, it can be then compared with MPM results. In other means the MPM reliability on dynamic modelling can also be checked.

Based on the gained knowledge from the verification phase, after the verifying both methods, a final ship-berm model will be created, simulated and the results will be compared.
CHAPTER 2: RESEARCH RELATED INFORMATION

2.1. PROTECTIVE MEASURES

For bridge piers different physical protective structures methods have been applied in various projects. A number of these methods will be shortly presented here while the main focus will be on the Protective berms or islands. These berms are also applicable for protection of tunnel approaches. Depending on the project requirements and the crossing ships, the decision will be made for the properties of a possible protective structure.

The design of the protective structures is mainly done by considering one of the following criteria:

- The whole kinetic energy, generated by vessel’s velocity, should be absorbed by the vessel itself. This energy absorption will result in elastic and plastic deformation of the vessel’s bow. Therefore the collision protective systems will be design with enough strength to resist the total impact force without any substantial damage to the protective systems or the structure.

- Protective system is considered the main energy absorber. The elastic and plastic deformations of these systems have to be able to absorb the impact energy without allowing any damage to the structure. In this criterion the damages to vessels is minimized.

- A combination of the two above mentioned criteria. The kinetic energy in this case is transferred into elastic and plastic deformations (beside generated heat) of both protective system and vessel such that no damage to the structure occurs.

To design the protective system, mostly, an iterative calculation is performed resulting in a trial initial configuration. A force vs displacement graph will then be created for the trial system using whether a physical or numerical model. The resulted force and energy capacity, which is estimated using the area under the curve, are compared to the design impact force and energy of the navigating ships. This comparison allows the designers to check the resistance of the protective systems. This should be considered that, in case of higher resistance of protective system than the ship impact force, the crushing of the vessel’s bow is the phenomenon that absorbs most of the impact energy. The other way around, if the protective system does not have enough resistance against vessel impact force, then the impact energy will be initially absorbed by the displacement and destruction of the protection system. In a case which both the vessel and protective system are being damaged and deformed, the percentage of allocated damage to the vessels has to be known in designs. This is a complex calculation therefore usually theoretical analysis, experience and physical modelling are all required.
Fender Systems
Fender systems are physical protective structure which tries to decrease the impact on bridge piers and/or force the ship into a different navigational direction. These systems are less suitable for head-on collisions because of their high elasticity and deflections. However they are widely used in rivers where smaller ships or bouts are navigating.

Fender systems are constructed using different methods and with diverse materials such as timber, rubber, concrete and steel.

Timber Systems:
A combination of vertical and horizontal parts made in grillage geometry are attached to the piers or created as independent structure in front of to the pier. Elastic deformations and crushing of the timber are responsible for energy absorption. Due to relative low cost and good energy absorption characteristics, timber fenders are mostly used for projects with relatively lower vessel impact forces. In case of large impact forces, based on the design vessels, significantly large timber fenders are required resulting in high cost and therefore not frequently used.

Timber fenders are also used in protective systems such as pile-supported structures and dolphins. They act as anti-sparkling in these cases and prevent metal-to-metal contact. In protective systems where steel elements, such as bolts and plates, are exposed this becomes of a high importance.

Rubber Fenders:
There are numerous ranges of choices for rubber fenders as protective systems. The energy caused by the colliding ship is absorbed by the elastic deformation of the rubbers and other fender system elements. This can be whether by compression, bending, shear deformation or a combination of all. Rubber may also be used in combination with other materials.

Concrete Fenders:
Concrete fenders are constructed in a hollow box like shape and are attached to the main structure. On the front face of these fenders, usually timber fenders are placed. The crushing or buckling of the concrete is the energy absorbing mechanism for these types of fenders. The energy absorption capacity for these fenders is in a direct relation with the wall thickness and the dimensions of the concrete box. The concrete fender needs to be fully replaced after a collision.

Steel Fenders:
Steel fenders consist of thin-walled membranes and bracing elements composed in a variety of box-like arrays and assemblies attached to the bridge pier. Impact energy is absorbed by compression, bending, and buckling of the steel elements in the fender. Timber facing should be attached to the steel fender to prevent sparks resulting from direct contact with steel-hulled vessels.

Polymeric Materials:
These systems are designed to be as flexible as possible (similar to rubber fenders) to absorb the energy. The flexibility of the system, regarding the maximum allowable deflection, is project dependent. This has to be designed in a way that the risk of any impact on the main structure to be minimized.
Fiberglass, fiber or structurally, reinforced with lumber composites are the used material and in case of hollow piles, concrete can be poured in to reach the desired strength and stiffness.
Pile-Supported Systems
Depending on the requirements and predicted impact forces, a group of piles that are connected with a rigid, sometimes flexible, pile caps are the energy absorber and collision preventive structures. These structures can be whether attached to the main structure, bridge piers, or be free standing. In any case they are required to stop the ships before reaching the main structure.

The energy absorbing mechanism in pile supported systems are either the bending in vertical piles or bending and compression of batter piles. To add extra resistance to the system, fenders can be attached to the pile caps.

Dolphin Protection
Dolphins are usually made in circular shape and most commonly pile supported. Embedded steel sheet piling, as outer material filled with rock or concrete, is another method for Dolphin construction. They also can be made of precast material and being floated to the position in water.

Design procedures for dolphins are usually based on an estimate of the energy changes that take place during the design impact loading. Energy-displacement relationships are typically developed for the following energy-dissipating mechanisms:

- Crushing/lifting of the vessel's bow
- Friction between the vessel and the dolphin or the river bottom
- Sliding, rotation or deformation of the dolphin

In order to prevent any sparks due to ship contact with the dolphin (metal-to-metal contact), timber or rubber fenders are used. This issue is a bigger concern in a case that of flammable products is present in a steel hulled vessel sailing through the waterway. Therefore the crossing vessel types need to be known for the design phase.

The dolphins are designed in a way that their shape will force the colliding vessels to change its navigational course and prevent a collision with the pier. Therefore dolphins should be capable of resisting a head-on impact forces. The energy is whether absorbed by crushing of the vessel's bow, the dolphin or a combination of both. In case of energy absorption by the dolphin, a large translational and rotational deformation will occur. It should be noted that since these protective systems are only active in case of a collision exactly at their location, they can only be considered as a protection for bridge piers (as a point construction) but are not convenient for immersed tunnels where a wider area (line construction) in in risk of a collision.

![FIGURE 3: DEFLECTION AND ROTATION OF DOLPHIN [1]](image-url)
In a case where both cell and vessel are contributing for energy absorption, large plastic deformations are usually acceptable. However, recommendations allow the maximum displacement of the top of the dolphin to be half of the dolphin diameter. The embedded sheet piles should be long enough to not being pulled out when the dolphin is rotating. The energy absorbing process will differ in each project based on the requirement. Therefore in the design phase, when a dolphin protective system is being used, a relation between the cost and the resistance of the dolphins is considered. As it is clear a larger dolphin with higher energy capacity results in higher costs. On the other hand, smaller dolphin may lead to damage to the main structure, decreasing the safety and increasing the project risk.

**Floating Protection Systems**

As the category name says, these systems use floating objects that are connected whether to ground or other structures to minimize the impact forces. Floating systems are mainly divided into three different categories.

**Cable Net Systems:**

This method uses the tension of cables to absorb the vessel energy. The cables are anchored to the seabed and a suspension system using buoys helps them to withstand the applied forces and energies.

**Anchored Pontoons:**

Large floating pontoons anchored to the seabed in front of the piers absorb vessel impact.

**Floating Shear Booms:**

Floating structures anchored to the waterway bottom that deflect aberrant vessels away from piers and absorb impact energy. In case of steel cables, chains and anchors for all methods, the corrosion has to be considered and prevented. The floating protection systems are mainly used in a deep water condition where the usage of other systems is not possible or requires a lot of money and time. As mentioned above the tension and deformation in the systems are the main mechanism to absorb the impact energy. The system can be design based on either elastic or plastic energy conversion.

**Island Protection**

Island protection, Protective Island or Collision berms is the main focus of this research. This method uses fill material around bridge piers, on top of an underwater tunnel or in front of the tunnel approaches to absorb the collision energy and prevent any contact or damage to the main structure, see Figure 4. As the main focus of this report will be on this method, collision berms and their properties will be explained in detail further on.

![FIGURE 4: ISLAND PROTECTIVE SYSTEM [1]](image-url)
Advantages and disadvantages
All of the mentioned methods have their own particular usages. The requirements are highly project related and aspects such as cost, energy capacity, characteristics of the design vessel, water depth, soil conditions and available space are of importance. Some methods are more suitable for adjusting the ship navigational course (Fender systems) and others, most importantly the protective berms, are highly effective for head-on collisions for heavier ships where other methods may not be capable to provide enough protection or resistance. Table 1 summarizes a number of advantages and disadvantages of these methods.

<table>
<thead>
<tr>
<th>Methods</th>
<th>Advantages</th>
<th>Disadvantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Timber Fenders</td>
<td>Low cost and good energy absorption relative to its cost.</td>
<td>Limited energy absorption compared to other methods.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Damage due to marine borers.</td>
</tr>
<tr>
<td>Rubber Fenders</td>
<td>Adequate energy absorption through elastic deformation.</td>
<td>Relatively higher initial cost compared to timber.</td>
</tr>
<tr>
<td></td>
<td>Compared to timber they have lower maintenance cost and higher durability.</td>
<td></td>
</tr>
<tr>
<td>Concrete Fenders</td>
<td>High (achievable) energy absorption capacity.</td>
<td>Difficulty in analyzing the structure's energy absorption characteristics while undergoing plastic deformation.</td>
</tr>
<tr>
<td></td>
<td>Resistance against natural and biological deterioration.</td>
<td>Needs to be fully replaced after an impact.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Possibility of steel corrosion through crack.</td>
</tr>
<tr>
<td>Steel Fenders</td>
<td>Can absorb high impact energies due to both elastic and plastic deformations.</td>
<td>Susceptibility to corrosion and metal to metal contact.</td>
</tr>
<tr>
<td></td>
<td>High feasibility in case of difficult seafloor conditions.</td>
<td></td>
</tr>
<tr>
<td>Pile Supported Systems</td>
<td>Can be designed with high energy absorption.</td>
<td>The damaged part needs replacement after collision.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Complex design.</td>
</tr>
<tr>
<td>Dolphin Protection</td>
<td>Prevents head-on collision.</td>
<td>Not suitable for side impacts and tunnels.</td>
</tr>
<tr>
<td>Floating Protection Systems</td>
<td>A good solution for deep waters where other methods are too costly.</td>
<td>Possibility of being overridden and corrosion of the system material.</td>
</tr>
<tr>
<td>Protective Berms</td>
<td>Easy and less expensive to construct with high energy absorption capacity.</td>
<td>Difficult to calculate.</td>
</tr>
<tr>
<td></td>
<td>Generates less damage to the ship hull.</td>
<td>Needs reconstruction after impact.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Lower feasibility in very high water depth.</td>
</tr>
</tbody>
</table>
An important aspect regarding the mentioned methods is that not all of them are suitable for both bridge piers and tunnels. The only method that could protect both constructions in case of a whether head-on or side collision is the Berm protective system as this report analyses and models.

Beside all these different protective measures, a number of different aspects have to be considered. Although these aspects are out of focus of this research, they are worth mentioning:

- For movable bridges an extra attention has to be paid for the electrical systems and cables. They need to be placed whether out of ships reach or be protected from any possible collision to be able to have full functionality all the time.
- Another tool to increase the overall safety of a bridge against ship or vessel collision is an extra warning system for the motorists and approaching ships. These systems may also warn the bridge crossing traffic, if collision is detected or anticipated, to decrease the possible casualties.
- Other preventive measures, such as alternative navigation routes for ships and navigation aids should be considered, where possible and feasible.

### 2.2. PROTECTIVE BERMS (COLLISION BERMS)

Collision berms or Protective berms are the main focus of this report therefore more detail information and explanations is given about them. These berms are considered more convenient for protection of both bridges and immersed tunnels as they can absorb high impact energy without any caused damage to the structure.

As briefly mentioned and visualized (Figure 4) in previous section, one of the advantage of submerged artificial berms as a protective system for offshore structures is their way of collision prevention. With a correctly designed and engineered berm, all large vessels will stop before reaching the structure; therefore any direct contact between colliding ships and the structure is prevented. Skidding of empty towed barges, due to their relatively lower weight, in case of high waters should not be neglected. Small draughts of sailing boats, small and medium yachts may allow them to travel more distance after hitting the berm however the residual impact force for the structure can be neglected in this case.

The main disadvantage of this method could be its influence on the environment. The berm can act as a blockage leading to changes in water current. It could also influence the marine life. Therefore this factor should not be neglected in a project.

#### 2.2.1. MECHANISMS OF THE BERM IN CASE OF A COLLISION

Collision of a ship with a protective berm has similarities with a grounding ship and same procedure leads to transfer of all ship’s kinetic energy. Berms can absorb the kinetic energy of the ship by a combination of friction and deformations within the soil.

In case of a collision, an expected mechanism is that the bow of a ship hits the slope, made from soil, and it will be forced to go upwards by the great amount of transient pressure created in the soil. At the beginning of the impact the ship will push the soil and force it to undergo localized failure. This mechanism is visualized in Figure 5.
After the impact while the ship is still travelling horizontally, the ship is pushing the soil. The soil deforms plastically as well as elastically absorbing and transferring the kinetic energy of the ship. As shown in figure by using the red arrows, the soil in front of the ship is being pushed forward (compressed) and the mechanism results in soil being pushed upwards as well. The total soil reaction force in this case is the summation of the normal forces perpendicular to the slope and the frictional forces parallel with the slope. When the maximum shear force capacity or resistance of the soil is reached, the soil particles will roll on top of each other. This is considered as a failure within the soil and the soil will separate (rupture) following a similar path as shown with red curves in figure. This phenomenon will continue with soil being gathered in front of the ship until the ship starts to slide rather than pushing through; or a combination of sliding and pushing will lead to stoppage of the ship. The ploughed soil in front of the ship will also contribute in energy transformation by sliding and friction.

Water pressure has also an influence on the total mechanism and displacement of soil and ship. The availability of pore water leads to ruptures within the soil effecting the lifting of the ship’s bow. A low permeable material can generate larger reaction forces in case of a collision forcing the ship to slide on top of the slope until the kinetic energy is transferred into heat by friction. So in a low permeable berm, less plastic deformations will be produced resulting in more needed distance to stop the ship, since most of the energy should be absorbed by friction. Soils with higher permeability stop the ship with a combination of reaction passive soil pressures (while deforming) and friction.

The availability of pore pressure can highly influence the total resisting force where according to test results given in literature [15], shown in Figure 6, the horizontal reaction force and the vertical lifting force are 10 to 20 times larger for a saturated and submerged berm than a dry berm.
In the Figure 6 the curves with a red circle at the end show the forces in case of a dry sand while other curves with blue circles are related to submerged slope. The tests are performed using different ship velocity (V). As it can be seen from the test results, in the submerged slope test both horizontal and vertical forces are significantly higher compared to dry slope, approving that the existence of pore water in the submerged slope and the generated flow by the impact can have a high influence on the movements and behavior of the ship after the impact. The increase in effective stresses in the soil that are cause by generated water flow results in both normal- and tangential-stresses on the ship. These stresses can be generated by two possible mechanisms:

- The suction forces caused by dilating sand in the rupture zones. These suctions result in increases in effective stresses within the soil skeleton.
- The increase in pore water flow caused by the compression of the soil squeezing the pore water out. The extra pore water flow generates large effective stresses around the soil grains.

By analyzing Figure 6, it can be seen that the squeezing of the pore water which generates a lot of excess pore water pressure under the ship bow, forces the bow to be lifted upwards. This means that the second mechanism is happening in this case.

2.2.2. **Analytical Calculation Method**

The numerical modelling of ship-berm collisions will be discussed in next chapter; however, an analytical calculation, based on the literature, is also explained in this section. The main parameter to be defined with this method is the minimum needed stoppage distance for the grounding ships.

Before proceeding to the mathematical calculations, the normative ship has to be defined. Therefore the design should be based on the maximum velocity, mass and size of the given ship.

American Association of State Highway and Transportation Officials (AASHTO) [1] has published a guideline in 2009 for analysis of the protective islands against ship collision. This report together with the provided technical report of Ramboll Arup TEC (RAT) for the Fehmarn immersed tunneling project, are used to perform a calculation for defining the requirements of a protective island.

The method is based on momentum and energy considerations. It uses the generated kinetic energy of the moving ship at the impact to estimate the required berm length in order to transfer the whole kinetic energy into heat by friction and potential energy by lifting of the ship. In other words, the ship undergoes the same procedure as it is being grounded. A grounding ship, beside the movement in its three planes, can also rotate around its three axes, see Figure 7. For a head-on collision of the ship with a berm, the generated friction energy is dependent on the ship movement in the x axis (surge). Vertical movement of the ship (heave) and a possible rotation around y axis (pitch) are the factors influencing the potential energy of the ship.
Calculations

The calculations are based on a head-on ship collision since this is considered as the most damaging collision between ships and bridge piers or tunnel approaches. This also allows for a 2 dimensional analysis. Further, in this method a number of important assumptions are made. First is assuming a fully rigid bow for the vessel which means that no energy is being absorbed by elastic or plastic deformation of the vessel’s bow. The next assumption is that the vessel is sliding on top of the soil with a constant contact area therefore no 3 dimensional effects are taken into account. Although these assumptions may not be fully realistic, however they allow the calculation to be only based on kinetic to friction and potential energy transformation.

Further the suggested calculation method is highly dependent on the chosen friction coefficient of the soil material.

According to the explanations above, the steps described below has to be followed to estimate the required stopping distance.

Step 1: Ship kinetic energy calculation:

For kinetic energy the straightforward formula based on velocity and mass is applied (eq. 1); however the added- or virtual-mass ($m_{add}$), the extra inertia caused by the amount of water which travels with the ship, should be taken into account. The virtual mass is added to the weight of the ship. This is defined mainly based on the shape of the ship and should be predefined before calculation.

$$E_{kin} = 0.5 \cdot (m + m_{add}) \cdot V_{ship}^2 \ [kJ]$$

where:
- $E_{kin}$: Kinetic energy [kJ]
- $m$: Mass [kg]
- $m_{add}$: Added mass [kg]
- $V_{ship}$: Ship velocity [m/s]

Step 2: Calculation of ship heave and pitch:

Since the vertical movement of the ship influences its potential energy, heave and pitch should be calculated. At this step the slope angle (defined as $\alpha$) of to be created berm should
be known or predefined. Based on this slope angle, the heave (non-definitive indicated with ‘*’) and pitch of the ship is calculated after each meter that it moves forward.

\[
Heave(i)^* = \frac{H_{bow}(i) + H_{stern}(i)}{2} \quad [m]
\] (2)

\[
Pitch(i) = \arctan\left(\frac{H_{bow}(i) + H_{stern}(i)}{L_{ship}}\right) \quad [°]
\] (3)

With:

\[
H_{bow}(i) = \frac{x(i)}{\cot(\alpha)} \quad [m]
\] (4)

For \( x(i) \leq \frac{2}{3}L_{ship} \) → \( H_{stern}(i) = \frac{H_{bow}(i)}{2} \quad [m] \) (5)

For \( x(i) > \frac{2}{3}L_{ship} \) → \( H_{stern}(i) = H_{stern}(i - 1) + \left(\frac{x(i) - \frac{2}{3}L_{ship}}{\frac{1}{\cot(\alpha)}}\right) \quad [m] \) (6)

where:

- \( i \): Calculations step \([-\]
- \( Heave^* \): Non-definitive ship heave \([°]\)
- \( Pitch \): Ship pitch \([°]\)
- \( H_{bow} \): Height of ship bow \([m]\)
- \( H_{stern} \): Height of ship bow \([m]\)
- \( L_{ship} \): Ship length \([m]\)
- \( x \): Ship horizontal movement \([m]\)
- \( \alpha \): Slope angle \([°]\)

It should be noted that the maximum possible heave is limited due to presence of the slope/berm. Therefore the final and definitive heave that will be used for further calculations is defined by:

\[
If \ H_{keel} = H_{crest} \rightarrow \ Heave(i) = Heave(i - 1)^* \quad [m]
\] (7)

\[
If \ H_{keel} \neq H_{crest} \rightarrow \ Heave(i) = Heave(i - 1) \quad [m]
\] (8)

With:

\[
H_{keel} = \min(H_{keel,max} + Heave, H_{crest}) \quad [m]
\] (9)

where:

- \( Heave \): Definitive ship heave \([°]\)
- \( H_{keel} \): Height of ship keel \([m]\)
- \( H_{keel,max} \): Maximum height of ship keel \([m]\)
- \( H_{crest} \): Height of ship crest \([m]\)

**Step 3: Vertical and horizontal impact forces:**

The vertical impact force is calculated by defining the vertical movement of the ship after impact and sliding on the berm. While the ship moves upward and a certain volume of the ship rises out of the water and rests on the berm, the gravity force is the vertical force which is used in this calculations.

\[
F_V(i) = Heave(i) \cdot w_{ship} \cdot L_{ship} \cdot \gamma_{water} \quad [kN]
\] (10)

where:

- \( F_V \): Vertical force \([kN]\)
- \( w_{ship} \): Ship width \([m]\)
- \( \gamma_{water} \): Unit weight of water \([kN/m^3]\)
Chapter 2: 
Research related information

For calculating the horizontal force, the relation between the perpendicular force on a surface and the generated friction force is followed. Therefore a crucial parameter is the friction coefficient ($\mu$). This value is predefined based on engineering judgment and experience.

$$F_H(i) = F_v(i) \cdot \mu \ [kN]$$  \hspace{1cm} (11)

where:
- $F_H$: Horizontal force \ [kN]
- $\mu$: Friction coefficient \ [-]

**Step 4: Energy dissipation:**
As mentioned previously, the kinetic energy of the moving ship will be transferred partly to potential energy and rest of it to heat due to friction. Calculation of the generated potential energy follows a straightforward formulation which is defined by the vertical movement of the ship.

$$E_{pot}(i) = m \cdot g \cdot Heave(i) + E_{pot}(i - 1) \ [kJ]$$  \hspace{1cm} (12)

where:
- $E_{pot}$: Potential energy \ [kJ]
- $g$: Gravitational acceleration \ [m/s²]

The dissipated energy due to friction is calculated by:

$$E_f(i) = 0.5 \cdot (F_H(i) + F_H(i - 1)) \cdot (x(i) - x(i - 1)) + E_f(i - 1) \ [kJ]$$  \hspace{1cm} (13)

where:
- $E_f$: Frictional energy \ [kJ]

Total generated energy is the summation of potential- and friction-energy.

$$E_T = \sum E_{pot}(i) + \sum E_f(i) \ [kJ]$$  \hspace{1cm} (14)

where:
- $E_T$: Total energy \ [kJ]

**Step 5: Final stoppage distance:**
Based on the energy generation by each meter that ship compresses the soil or slide on the berm, at the point where the total energy $E_T$, calculated in step 4, is equal or bigger than the initial kinetic energy of the ship, that distance is considered as the final stoppage distance of the ship.

**Issues with this calculation method:**
- The most important and crucial issue with this method is its lack of dependency on soil parameters. The different parameters of soil are not directly influencing the calculation; only the friction coefficient between soil and ship is taken into account which in a way can be correlated with internal friction angle and dilatancy angle. Other parameters such as soil stiffness, cohesion and permeability are not taken into account. These parameters have significant effect in reality.
- The contact angle, the ship does not always have the same slope angle as the berm, at the beginning it might be just a small area (due to presence of a bulb) and increasing in time.
- The soil is deforming therefore changes in resistance can appear. Since it is being compacted, resistance could increase.
• The contact area is changing by the ship displacement in any direction. At the beginning it might be just a small area (the ship bow) and increasing in time. Increased contact area results in a more friction thus ship will stop in shorter time.

• As the ship is proceeding forward and moving into the berm, the soil in front of it will be pushed up and/or aside. After some time the risen up soil will come in contact with the ship influencing the total resistance.

• The stress and force propagation are not mentioned and calculated. Therefore no indication of the forces on main structure is obtained.

Possible improvements:
As stated above one of main shortcoming of this analytical method is that the stiffness, strength and permeability of the soil is not directly influencing. The strength of the soil is included in terms of surface friction.

The surface friction or the interaction between the ship surface and the berm can be defined using a relation between surface friction \( \mu \) and internal friction angle of the soil \( \varphi \). The value of \( \varphi \) is a known soil parameter.

If the soil is made of a uniform layer the surface friction can be taken equal to tangent of internal friction angle. However in engineering world and based on experience the frictional resistance of a soil is estimated as \( \frac{2}{3} \) of the internal friction angle of the soil.

\[
\mu = \frac{2}{3} \cdot \tan(\varphi) \quad [-]
\]

where:
- \( \varphi \): Internal friction angle \([\degree]\)

Using this value provides a safer margin while calculating and it can be implemented in the analytical method resulting in a more accurate frictional resistance of the soil.

It should be noted that the equation 15 uses the internal friction angle defined for a drained situation where no pore water pressure changes are available. A ship-berm collision acts as an undrained problem where pore water pressure between the soil grains alters their shear resistance. A value that is obtained from an undrained test, the consolidated-undrained angle of shearing resistance, could also be used here; however using this value brings complexity that cannot be checked or defined without specific tests. The excess pore water pressure is considered as a critical parameter which cannot be easily estimated in such a problem. Therefore by knowing the fact that the whole analytical method provides an indication of the total stoppage distance, the internal friction angle is suggested to be used in order to keep the calculations doable.

Due to the fact that the main used fill material for the berms is sand which is a permeable material, it is worth mentioning that other factors will influence the total strength and stiffness of the berm and by being taken into account in the calculations, they will improve the estimations. Some of these factors are:

• Density of the sand
• Normal stresses
• Grain size and shape
• Dilatancy of the sand
• Existence of pore water pressure
• Stiffness of fill material
2.3. **FINITE ELEMENT METHOD (FEM)**

Finite Element Method (FEM) is a well-known and widely used numerical method in all specialties. Therefore throughout decades of experience and experiments the theory of FEM has been highly developed. This method provides good approximations for space- and time-dependent problems that are difficult to calculate analytically. The approximations of partial differential equations can be done by using different type of discretization. For steady state problems, algebraic equations using numerical linear algebra methods are applied; while Euler's method, the Runge-Kutta or Newmark methods are the common numerical integration techniques for transient problems.

When FEM is being used the problem is divided into smaller parts, finite elements, and then the equations that are modelling these parts are assembled together representing the whole problem. Mesh generation techniques are used for this matter where the elements are connected at common points for two or more elements with boundary lines or surfaces.

The method provides estimation for an unknown at chosen points within the problem domain by small changes in independent variable such as time. The error estimation is also important since the convergence of the estimated unknown occurs when the error tolerance has been reached.

In FEM, the values at non-nodal points are approximated using the boundary variables computed at element nodes by interpolation of the nodal values. The approximation is done using the interpolation functions also known as shape functions.

The procedure of performing an analysis using FEM can be divided into:

- **Preprocessing**: in this stage the following is defined:
  - The geometry of the problem
  - Type and material properties of the elements
  - Interconnection and connectivity of the elements
  - Boundary conditions and loads
- **Results**: where the unknowns are computed and are used further in back calculations of additional derived variables such as reaction forces, stresses and heat flow
- **Post processing**: the stage where all the needed parameters and information are computed and needs to be presented whether as just values or be plotted.

2.3.1. **FEM CHALLENGES AND ISSUES**

One of the limitations of regular FEM software at the moment is the fact that they cannot handle geometrically nonlinear-large deformation problems.

Despite all the experience and development on computational techniques, modelling and analysis of problems such as landslides where large strains are available is still a difficult task. FEM does not offer enough robustness for these types of problems since the distortion of elements and mesh lead to inaccuracy or even causing the calculations to stop due to negative values of Jacobian determinants at integration points. A possible improvement to overcome this issue is the re-meshing technique; usage of this technique leads to another issues like additional errors due to projection of the results from a distorted mesh to the new mesh.
The main challenge that is related to this research topic in FEM is the dynamic movements and consequently the large deformations in the ship-berm problem. In static analysis in case of a large deformation the calculations stops, as mentioned above, and the soil is considered to be collapsed and failed. Another big challenge regarding this topic is the fact that the contact area is constantly changing by any movements of the ship and the deformation of the soil. This adds to the difficulty of the formulations and modelling.

On the other hand, current FEM software does not offer an option to allow an element to move through the space; and elements should also be connected in a way in order to have any interaction. Therefore creativity and experience is required to create a reliable working dynamic model.

2.4. MATERIAL POINT METHOD (MPM)

MPM is a relatively new method and is considered as a useful and suitable method to be used for calculations and analyses of fluid dynamics and problems with possibility of large deformations. This method has been introduced by Sulsky et al. (1994) for the first time for modelling solid materials. Since then MPM is being further developed by researches to become more applicable in various problems. In recent years more focus has been put on MPM for geotechnical applications especially in Offshore engineering where more deformations and displacements are expected. These applications are still more research based rather than practical usage.

In geotechnical engineering and problems that are relevant to soil, Finite Element Method (FEM) had been used through the past decades. In case of large deformations and material flows within soil body, FEM has some limitations. For these types of analysis, Material Point Method (MPM), an alternative to FEM is developed. In MPM the information such as stress and deformation are stored in material points and therefore despite existence of a background mesh, MPM does not have the limitations of a mesh-based method. In other words, the Lagrangian particles in this method are allowed to move through the Eulerian mesh. In large deformed models, the Eulerian mesh, after each calculation step, is reset to its original position but the displaced particles, or material points, stay in updated form and position. This is considered as one of the main advantages for models with large deformations since material points are allowed to move through the grid.

Aside from the advantages of MPM compared to FEM, MPM has its own issues and drawbacks. MPM is proved to be more expensive and requires more storage space since it consist material points and a mesh as well. Furthermore the resetting process of the grid after each step and the initializations for the next step takes more time.

MPM models do not necessary require a high end system to be simulated. The simulations can be performed using an above average system. However the required memory and storage capacity is highly dependent on the mesh size, i.e. the number of elements and consequently material points within the model, and number of computation steps made in a calculation.

MPM models may result in less accurate calculations due to the possibility of the oscillations when the particles are crossing the mesh boundaries. The strain localizations in MPM are also more dependent on the mesh size and orientation.
2.4.1. INTRODUCTION TO MPM FORMULATION

MPM can be considered as an update for the FEM. Therefore the basic of MPM has the same formulation as FEM. In MPM the material points which are the integration points, are allowed to move through the grid instead of the element edge. MPM could be divided into three phases [3]: the first two phases, the Initialization Phase and Lagrangian Phase are following the same principle as FEM while the last phase, the Convection Phase where the mesh is restored with material points staying at their current new position, makes the difference between two methods (Figure 8).

![Figure 8: MPM Calculation Phases](image)

The different steps and phases regarding an implicit formulation of MPM, based on Brinkgreve et al. (2015), are briefly explained below:

- The conservation equation for linear momentum is the governing equations for continuous bodies in MPM; After a transformation of the strong form of equation into a weak form, the standard FEM procedures is being used for discretization which is then multiplied by finite element shape functions
- For solving the equation of motion in MPM for each calculation step, the Newton-Raphson method which is the same for FEM is applied; and the calculations continues until the error is smaller than the defined requirements. In MPM discretization of time derivative which allows the calculation of acceleration terms is done by a trapezoidal rule
- The standard shape functions are used to interpolate the state variables that are stored in material points as initialization of each calculation step, to the computational grid

The computational grid is considered as the current configuration of the model while the Updated Lagrangian formulation of discrete equations is used to compute the increments in displacement. The results deform the computational grid therefore the kinematics of the system has to be updated before the next calculation step. When the calculated error has met the criteria, the state parameters of the material points are updated from the computational grid. This procedure uses the standard approximation functions that are defined on the mesh, and is performed by interpolation of the nodal results from the grid to the material points.

With this method of calculations, after computation of the variable at material points, the deformed grid can be discarded, since all the information are already stored in the material points. This prevents any distortion of the mesh.
2.4.2. **CHALLENGES AND ISSUES**

A common issue of MPM, according to the paper by Gonzalez Acosta et al. (2017) [7], is the possible stress oscillations and inaccuracies. These issues are originated from the shape function approximations and have two possible reasons. Firstly, the inaccuracy can be caused by numerical integration using the material point method that derive the component of stiffness matrix; secondly, calculation of the strain from the nodal displacements of the elements. In the same paper a thick-walled cylinder with an internal pressure has been analyzed in order to present the possible stress oscillations in MPM. The analysis has been performed for linear and high order elements. The stress distribution, resulted from the analysis is presented in Figure 9 with 'p' the applied internal pressure and ‘σ_r’ the radial stress on the vertical axis.

![Figure 9: Stress Oscillation for 4 Noded Elements (Left) and 8 Noded Elements (Right)](image)

The graphs show that the stress distribution has a different shape and magnitude for different elements and is dependent on the Poisson ratio value (v). The higher order elements result in a second order stress distribution (Figure 9 right) while an almost linear stress distribution is obtained for a linear element (Figure 9 left). Also, it can be observed that for a linear element more stress oscillations are occurring. These oscillations increase at the boundaries due to the fact that the shape functions are not continuous anymore and the material points are crossing the inter-element boundaries.

To mitigate the stress oscillations in MPM, the paper by Bürg et al. (2017) [4], presents the Dual Domain Material Point (DDMP) which is developed from derivation of second-order MPM. This update results in a smoother stress distribution throughout the whole domain. Modified shape functions are suggested in DDMP which removes the possible instability and oscillations if the material points are crossing an inter-element boundary.

In MPM the material points are free to move within the model. These points have to be inside an element in order to keep and process the information on them. To tackle this issue different approaches have been made in different MPM formulation. The made approach by Plaxis MPM is the usage of predefined dummy cells/elements. These elements have to be placed at locations where the material points are expected to move and exit the active mesh in a simulation. For these cells an elastic stiffness is applied to avoid singularity in equations due to zero stiffness or mass contribution in global matrix. This also applies if a cell becomes empty due to deformation and leaving material points.

Compared to FEM, MPM is more time consuming which is dependent on many aspects. A possible solution is to divide the model into different domains; where less deformation is
expected the FEM domain could be applied and MPM domain for large deforming zones. Therefore the problem and model should be very well pre analyzed to reduce the modelling size and calculation time. The required storage space can also be reduced by this solution.

Using a combination of FEM and MPM in a model introduces another challenge for the formulations which is the connection between the FEM and MPM domains. The integrals in FEM are calculated using the conventional quadrature points but in MPM the material points are used as the quadrature point. In an implicit MPM formulation this coupling is being done automatically within the calculations. The main difference here is that in MPM, in addition to the convection step, after each calculation step, the mesh relaxation procedure has to be performed in order to restore the deformed mesh.

Another important aspect in MPM modelling is the element contact formulation. For element contacts in MPM, rigid contact is included in the formulation, however these contacts do not provide realistic and accurate results if used for soil-structure interaction. A level-set large sliding algorithm have been applied in Plaxis MPM to tackle this issue [3]. In this algorithm, which has been introduced by Anreykiv et al. (2011), two different meshes are used to model the two contacting elements.

In general making the MPM calculation fully stable is a difficult task and procedure. All the additional required steps and formulations, compared to FEM, together with boundary conditions, empty cells treatment and material points per cell, cause extra difficulties and challenges in this method. Therefore it requires a lot of knowledge. Nevertheless its advantage in certain application compared to FEM, compensate for its challenges.
CHAPTER 3: NUMERICAL MODELLING

Numerical modelling is the main part of this research. This chapter presents all the steps taken to reach and create the final ship-berm model.

3.1. CHECKS AND VERIFICATIONS

The verification and checks are performed on the simpler models that are the basis for the final model which include elastic-, plastic-deformation and dynamic movements. These checks and verification are necessary and important, especially for MPM, due to the fact that the used Plaxis MPM software is still under development and not a finished product.

Further by verifying the results from these models it can be observed whether the final model is reliable or not. The FEM models are to be checked initially and then compared with the MPM results leading to verification of the MPM models.

The model that includes both plastic and elastic deformations of the soil body is being verified first. This model considers the Mohr Coulomb (MC) criterion which is a well-known approach in Geo-Engineering applications. Using MC criterion allows for having a good comparison between FEM and MPM when plastic and elastic deformations are both included.

The second check is performed on a model which simulates a dynamically moving object.

3.1.3. MOHR COULOMB MODEL

Due to the fact that it is a difficult task to calculate the plastic strains without numerical modelling, the validation of Elasto-plastic models (material parameters based on Mohr Coulomb) follows a different procedure. The formula, according to Mohr Coulomb (MC) and Prandtl criteria, gives a direct relation between the applied load and the shear strength of the soil in a soil model shown in Figure 10a. The model consists of a soil with predefined undrained shear strength and an infinite depth so that no bottom boundary effect is available. At the sides the boundaries are only allowing vertical displacement and no horizontal movements. A line load is applied on top of the soil and spread over half of the total width. In plastic Plaxis FEM calculation in, at the point where the soil collapses the load can be checked; In other words, the maximum load at the point where displacement is increasing while load stays the same.
Chapter 3: Numerical modelling

The soil deformation on top of this model, where the load is being applied should follow the same path but the opposite direction as the other half of the soil. This means that the displaced soil under the load is equal to the amount (in terms of volume) of the soil being pushed upwards, see Figure 10b.

It should be noted that in both FEM and MPM models, the displaced volume may not be hundred percent equal at both side. This is due to the fact that soil is not considered as fully incompressible leading to small compressions. However this does not have significant effect on the failure load.

The maximum displacement, beside the applied load, is also dependent on soil parameters such as stiffness and soil unit weight. Following parameters are used in both FEM and MPM models:

- Half of the width $a = 1$ [m]
- Line load $q = 550$ [kN/m]
- Undrained shear strength $S_u = 100$ [kN/m$^2$]
- Stiffness $E = 20000$ [kN/m$^2$]
- Unit weight = 15 [kN/m$^3$]
- Poisson ratio $\nu = 0.49$ [-]

The deformation in this model follows a similar path to the zone III of Prandtl's schematization for a strip load presented in Figure 11.

For a constant stress in figure above a possible stress field is shown in Figure 12 with $c$ the cohesion of the soil.
In case of a frictionless soil (the chosen soil in this model) the cohesion $c$ will be equal to the undrained shear strength. Therefore based on the Prandtl’s solution the vertical failure load $q_{\text{max}}$ can be calculated using:

$$q_{\text{max}} = (2 + \pi) \cdot S_u = 515.16 \ \frac{kN}{m^2}$$

The presented formula considers an analysis performed based on total stress analysis. Therefore it can be applied in a situation where a large deformation is available. However if the analysis is being performed based on effective stresses, the amount of the soil that has risen up will act as an extra overburden pressure (Terzaghi bearing capacity) influencing the whole failure mechanism. Since the presented analysis will be based on total stress situation, the used formula is applicable.

Figure 13 represents the result from FEM plastic calculation. Since FEM has already proven to be trusted and reliable for plastic or static soil calculations, as expected, at the point where load reaches a value of 514.14 [kN/m] the soil is collapsing. This can be seen by increases in displacement with a constant load.
Therefore the same model with same parameters has been created using the Material Point Method with results presented in Figure 14.

![Displacement vs Load](image)

**FIGURE 14: FEM AND MPM ELASTO-PLASTIC MODEL RESULT**

From the results it can be seen that both methods are reaching almost the same failure load. However the path is not totally matching. This can be due to a number of factors such as difference in formulation where the variables are computed. The calculations in FEM also are performed more continuously compared to MPM which is visible by looking at the graph.

Neglecting the displacement vs load path, both methods provide the failure load which is close to the expected value based on eq. 16. This confirms the verification of both methods.

### 3.1.4. DYNAMIC MODEL

The Finite Element Method (FEM) is mainly used for static calculation when it comes to Geotechnical applications. Availability of the dynamic calculation option in Plaxis 2D software allows the user to create dynamic models up to a certain level. This option includes the variation in velocity of an object. However this does not mean that an object can move freely in space before reaching an element. Due to less usage and relatively less experience with dynamic calculations, a simple dynamic model is created and the results are checked and verified.

In order to verify the results from Dynamic FEM calculations, a fully elastic model has been assumed as a starting point. By choosing a Linear Elastic (LE) model, and with a model that has a 1 m width, a simple check can be performed with the elastic deformation of a 1D rode element using analytical solutions.

The FEM model is made of a fully rigid falling cube with an area of 1 m², which is being dropped from 19.42 m, equal to a free fall distance after 2 seconds (approximately 1.99 seconds to be more specific), on a 13 m long, 1 m wide linear elastic material. The block will reach a velocity of 19.52 m/s (=1.99 x 9.81) right before the impact. A visualization of the model with the used parameters is presented in Figure 15.
In this model, in order to have the interaction between the block and the soil, a linear elastic material is created (19.42 m long) between the block and LE material with zero stiffness and weight. This connecting layer allows the kinetic energy and the impact of the block to be transferred to the soil while having no effect on any parameter.

The advantages of using a linear elastic model at this stage, since the model has a 1 m width, is that the model can be assumed as a one dimensional rod element which is being hit by an object. Therefore the straightforward relations between force, area, stiffness and strain (eq. 17 and 18) can be used to check the values from Plaxis model.

\[
\sigma = \frac{F}{A} \quad [\text{kN/m}^2] \quad (17)
\]

\[
\varepsilon = \frac{\sigma}{E} \quad [-] \quad (18)
\]

where:
- \( \sigma \): Stress [kN/m²]
- \( \varepsilon \): Strain [-]
- \( F \): Force [kN]
- \( A \): Area [m²]
- \( E \): Stiffness [kN/m²]

The main idea behind this model and approach is based on the conservation of energy. This means that the gained kinetic energy by the object will be transferred in potential energy which is equal to the elastic deformation of the material. Based on this theory, it can be said that at the point where the block has zero velocity, the whole kinetic energy has been transferred into the soil. At this exact time, the maximum displacement (compression) should be reached.

Figure 16 is the result taken from Plaxis. The presented results are starting from the moment that the block is touching the soil. Therefore at time zero the velocity is equal to 19.52 m/s where the displacement indicates the displacement of top of the soil which will be identical to the block after the impact.
As it is demonstrated with the green dotted line Figure 16, after 0.15 seconds the object reaches its maximum displacement (= -0.743 m) before bouncing back. The velocity is indeed equal to zero at this time step indicating that the kinetic energy is equal to zero.

The next step after confirmation of the energy transformation is to calculate the applied force by the falling block on the soil. This is not an easy task in general, however by knowing the time or the distance which the force is being applied, the force can be back-calculated based on work-energy principle.

The work done by the falling object on the LE material is equal to the force being applied over a distance. Beside the force that is resulted from dynamic movement of the object, the gravitational force has to be taken into account. The gravity force is working downwards (opposite direction of the resisting force F). The negative signs are an indication for upward directions for vectors

\[ W = (-F + mg) \Delta L \ [kN.m] \]  

where:
- \( W \): Work [kJ] or [kN.m]
- \( \Delta L \): Compression [m]

By assuming that energy is equal to the work done and the velocity is equal to zero at the end, the force is back calculated.

\[ \Delta W = \Delta E_{\text{kin}} = 0 - \frac{1}{2} mV^2 \ [kN.m \ or \ kJ] \]  

where:
- \( \Delta W \): Changes in work [kJ] or [kN.m]
- \( \Delta E_{\text{kin}} \): Changes in kinetic energy [kJ] or [kN.m]

Combining equations 19 and 20 gives:

\[ F = m \left( g + \frac{V^2}{2 \Delta L} \right) \ [kN] \]  

(21)
Chapter 3:
Numerical modelling

With $\Delta L = 0.743\ m$ the force is equal to $2715.5\ kN$. 

$$F = 10,200 \left( 9.81 + \frac{19.52^2}{2 \cdot 0.743} \right) = 2715.5\ kN$$

The verification phase follows a different approach. As mentioned above the relations between strains and stresses are used here to calculate the force. By assuming a rod element the total vertical strain $\varepsilon_v (=\Delta L/L)$ is equal to 0.05715. The stress can now be calculated and with an area of $A$ equal to 1 $m^2$ the force is back-calculated.

$$\sigma_T = \varepsilon_v E \left[ \frac{kN}{m^2} \right]$$

$$\sigma_T = 0.05715 \cdot 50 \cdot 10^3 = 2857.7\ \frac{kN}{m^2}$$

where:
- $\sigma_T$: Total stress [kN/m$^2$]
- $\varepsilon_v$: Total vertical strain [-]

$$F = A\sigma_T \left[ kN \right]$$

$$F = 1 \cdot 2857.7 = 2857.7\ kN$$

When comparing the two calculated values for impact force, it can be seen that the stress-strain relation results in slightly higher force, less than 5% difference. This can be due to the fact that in the numerical model, existence of the integration points has influence on the calculation. Therefore more force will be required to compress the material than the force calculated using analytical method which does not include any interaction within the material. However the small difference suggests that the model gives reliable results and is verified.

**Vertical versus Horizontal dynamic model**

The discussed dynamic model in the previous section covered a vertical dynamic problem while the final model requires horizontally moving object to be able to model the ship impact. Since the vertical model is already verified, the horizontal model will be compared with the vertical model. The vertical model used the gravitational force to bring a certain velocity to the object before the impact. This force is not available for a horizontally moving object. In the used FEM software, Plaxis, there are three possible ways to apply kinetic energy on an element, namely by applying velocity, displacement or acceleration on the element. Therefore all three are checked and compared in order to understand differences. These performed checks can be found in appendix A including the checks for keeping the element vertically in place and overcome the gravity force.

After running all needed checks, the same vertical model, with same properties, has been tilted and transferred into horizontal model. The moving block will also have the same kinetic energy at the impact. The results are compared and represented in Figure 17.
The represented comparison shows that the results obtained from both horizontal and vertical models are almost identical with slightly larger values for both vertical displacement and velocity after the impact. The extra gravity force that is all the time available in the vertical model could be considered as the main cause of this small difference. The whole soil body is being under influence the whole time leading to less resistance against the falling cube.

**Sensitivity analysis**

After creation of a dynamic model in horizontal direction, it is important to perform a sensitivity analysis on the model. By doing so, it can be confirmed that the model is working properly. For this case the most important parameters are the stiffness of the soil and the impact energy which is dependent on the mass and velocity of the moving block.

Therefore a sensitivity analysis has been performed by changing the soil stiffness, block mass and velocity. The results, which are all in accordance to the expectations, are presented in detail in appendix B with the following as a short summary:

- A soil with higher stiffness results in less compression and a faster decrease in block velocity while lower soil stiffness gives the exact opposite results. Here in case of a higher stiffness, the increase in stresses within the soil is the phenomenon that takes the kinetic energy of the block which is not transferred in elastic deformation; while more compression (for lower stiffness) leads to lower values for stress in soil
- Increases in kinetic energy of the block, obtained by whether a larger mass and/or higher velocity, increases the compression while lower kinetic energy leads to less compression

![Velocity & Displacement vs Time](image-url)
MPM Dynamic model
The next step is to check and verify the same exact dynamic model using Material Point Method. In MPM, unlike FEM, the object can freely move in space and no connection or initial contact is required between the block and soil. Therefore certain acceleration can be applied to the block, allowing it to reach the desired velocity over a distance. By pre-calculation of the required travel distance in certain time step, the desired velocity and consequently the kinetic energy can be achieved right before the impact. For a good comparison between MPM and FEM, three models with same geometry but different parameters are tested. All three MPM models show comparable or nearly identical results with explainable differences. Figure 18 and Figure 19 present the result of one of these models while other two are shown in appendix C.

![Soil Displacement vs Time](image1)

**FIGURE 18: MPM AND FEM SOIL DISPLACEMENT**

![Block Velocity vs Time](image2)

**FIGURE 19: MPM AND FEM BLOCK VELOCITY**
Due to the fact that in MPM the block and soil are not connected (or glued), two separate graphs are needed to be able to represent the displacement and velocity for MPM. One is showing the displacements of the soil and the other the velocity of the block; while in FEM, since from the exact moment of impact the block and soil are glued together, same values for displacement and velocity are obtained.

Figure 20 visualizes the phases of the MPM dynamic model.

---

**FIGURE 20: MPM DYNAMIC MODEL**

So in MPM the block will compress the soil until the whole kinetic energy is transferred into potential energy and the soil reaches its maximum compression with the block having a velocity equal to zero. While the elasticity of the soil is decompressing the soil and trying to bounce the block back, in MPM it can be seen that the block is being push until is fully detached from the soil. After being detached, since no resistance is available, the block continues with a constant gained velocity, see Figure 19.

By looking at Figure 18, the slight difference between the oscillations in MPM and FEM is due to the fact that in FEM, the connectivity of the block increases the resistance when the soil is extending and compressing again.

Aside from the logical differences in both models, the results are fairly close to each other in all three test models confirming the trustworthy of MPM.

### 3.2. Finally Model

Finally after all the necessary taken steps and verifications, the final models using both FEM and MPM can be created. This section presents both models, which are a combination of the simple models with their findings. The detailed results of the final models will be discussed in chapter 4.

#### FEM Ship-berm model

The final model is created from a berm with a total length of 60 m and MC material. So both plastic and elastic deformation will be considered. The slope of the berm has a ratio of 1:3 with 9 meter height. The soil has a stiffness of 30000 kN/m², a unit weight of 20 kN/m³ and an internal friction angle of 30°.
Although in reality the front side of a ship may not be a continuous line, due to existence of a possible bulb at the bow, for modelling the ship the best possible shape has been chosen which a continuous line with a constant angle is. Due to the fact that the contact area has to be predefined and will not change throughout the whole calculation; the front of the ship has the same slope ratio as the berm. If this was not the case, the contact would become just a point. A fully rigid material is chosen (elastic stiffness equal to $1 \times 10^{10}$ kN/m$^2$) for the ship with a total mass of around 2000 tons and a base length of 22 m. With a rigid material no energy is lost due to ship deformation. Figure 21 visualizes the FEM ship-berm model.

This model consists of 2 phases, the first phase is where the ship has no contact with the berm, or the berm is not active in simulation, allowing the ship to reach the desired velocity. In this phase the vertical movement of the ship is restricted while it is free to move horizontally. The ship is modelled to reach a velocity of 10 m/s, which is gained over a long time to have a uniform velocity. Once the desired kinetic energy is reached, in second phase, the berm is then activated creating the interaction between the ship and the berm. The model boundaries have been chosen in a way to dissipate the propagated stresses and do not bounce them back; in Plaxis FEM this can be done by choosing a viscous boundary option.

In order to simulate the sliding phenomena, an interface between ship and berm is created. This is shown by a diagonal light blue line with a minus sign underneath.

Interfaces in Plaxis are used in order to improve soil-structure interaction in a model. They are mainly modelled using a bilinear Mohr-Coulomb model [18]. In the zones where interface is created only relevant data (cohesion, stiffness, Poisson’s ratio, dilatancy- and friction-angle) are taken into account regardless of the complexity of the used soil models. Therefore the interface stiffness will be considered to be equal to the elastic soil stiffness. Interfaces can simulate, for example, extreme shear stresses in a thin zone which could not be done in a normal contact surface. Since in the ship-berm model high shear stresses are to be expected at contact surface, usage of this option is will help to improve the results.

Since in reality, the buoyancy force is keeping the ship from falling down, in this model a line load is created to represent this force. This might not be the ideal way of force recreation since by any vertical movement of the ship, the buoyancy force is changing, but is a good approach and simplification in this case.
Chapter 3:
Numerical modelling

MPM Ship-berm model
The MPM model is basically the same model as FEM with some, MPM specific, changes. Due to the encountering of all explained challenges in chapter 2, the modelling of MPM was more challenging and time consuming. The created model in the software interface could not be presented in this report due to confidentiality. However in order to be able to have a fair comparison between models, it has been tried to keep everything the same as much as possible. Therefore the MPM model has the same material and properties. The geometry is almost identical with only an increase in the total berm length to 100 m. This adjustment cancels the influence of any bounced back stresses since the boundary is at a greater distance. The boundary conditions and vertical movement restrictions have also been kept the same as FEM model.

In MPM the possibility of having a continuous changing contact area does exist. This means that after impact, by displacement of the ship and deformations of the soil, the ship-berm contact area is constantly changing. This big difference at this point can already show an advantage of MPM compared to FEM. In order to activate this phenomena and interaction between the moving ship and the berm, using the software interface, contact boundaries (as shortly explained in paragraph 2.4.2) have to be created at the areas that expected to be in contact with other elements. For this model the whole bottom part of the ship, the diagonal and vertical frontal part are chosen as contact boundaries. While creating a model with activated contact boundary, the stiffness difference between the contacting elements should not be extremely large. This could lead to oscillations of velocity and acceleration. The existing stiffness difference within the created ship and berm in this model is considered to be within the margin and did not cause any issues.

An important factor in MPM model is the mesh size and number of elements. With a finer mesh the number of material points or integration points is also increasing. A very fine mesh may positively influence the accuracy in some cases while increasing the required storage and calculation time dramatically as a downside. However a fine mesh does not always mean that a higher accuracy can be reached. Therefore an unnecessary fine mesh can only increase the required space and simulation time. For this model a medium sized mesh found to be ideal.

Regarding the mesh, a good method for gaining the best result, and avoid unnecessary simulation time, is to create an uniformly distributed mesh with a combination of FEM- and MPM-materials; where large deformations are to be expected, the MPM material should be used and at the locations that less deformation will occur, FEM material to be chosen. In the created model the same approach has been applied; and in order to generate a uniform mesh, the berm has been divided into three horizontal layers with equal heights.

While modelling in MPM the user should also have a good prediction for the possible soil deformations and displacement of material points. In Plaxis MPM if the points are expected to move, there has to be a dummy mesh available. The material points cannot move to an empty space. The presence of these dummy clusters should not have any influence on the results, therefore if they are not included within the model itself, should be deselected while performing the calculations. As an example, for the ship-berm model, two dummy clusters have been placed; one above the berm where the soil is expected to be pushed upwards and the other one in front of the berm where the soil may become loose and displace.

Aside from the mentioned points, in the MPM model there are a number of aspects that have to be considered:
• Not all the parameters used in the model are a good representative of the real world situation (applies also for FEM). This is due to the fact that the MPM software has not been fully developed to be able to handle the extreme impact forces of a large container ship. The weight of the ship is limited to 2000 tons which is a good representative of small to medium sized ships but not the big container ships. However the parameters within the soil/berm are chosen as realistic as possible.

• An artificial buoyancy force is applied to the ship to keep it from falling during simulation. The issue here, same as FEM, is the fact that this force will stay through the whole calculation procedure resulting in a reduced vertical force on the berm.

• Due to long calculation time, the size of the berm is restricted to a height of 9 meter, which is a good representation for tunnel approaches but it may vary depending on the water depth.
CHAPTER 4:
MODELLING RESULTS

4.1. RESULTS
The result of FEM and MPM model are presented in this chapter. The outputs of the methods are presented in a different way. The FEM model is shown using the Plaxis 2D output window while the MPM result are given by screenshots taken from visualization software (ParaView) since its own output section was not available.

4.1.1. FEM RESULTS
Figure 22 presents a visualization of the deformed mesh in FEM simulation after 1 second of calculation. The red line indicates the constant contact area with the dotted line showing its initial position.

![Figure 22: FEM SHIP-BERM MODEL RESULT (TRUE SCALE)](image)

By passing each calculation time-step the ship is moving forward, from left to right in the figure, and is deforming the soil (or the mesh) more and more. Since the contact area is staying constant an unrealistic phenomena is occurring. This is due to the fact that in Plaxis FEM formulation the new contact area, or the contact area changes, is not recognized. As a result the same exact area that initially connects the bottom side of the ship with the berm is being pulled away. This is not the case in real world problem. The pulling effect forces the mesh to undergo a deformation that is neither expected nor realistic. The piled up soil in front of the ship is going through the ship, as if the ship does not exist, while having zero effect on it.
By looking at the ship displacement and velocity, Figure 23 and Figure 24, it can be seen that the whole berm is not coming into interaction with the ship as the velocity is following an almost linear path, in other words the velocity is decreasing with a nearly constant negative acceleration. This indicates that no significant changes are being applied from the whole berm on the ship. The displacement graph suggests the same phenomenon since it follows a nearly perfect curved path with no oscillation in between.

Nevertheless, some useful information can be obtained from FEM results showing a possible soil failure path. This is somehow in accordance to the discussed failure mechanism previously and is visualized in Figure 25.
Figure 25 is showing the shear stresses ($\sigma_{xy}$) within the soil. The coloring scale in this figure starts from $-50$ kN/m$^2$ (dark blue) to $80$ kN/m$^2$ (red). The increases in the shear stress in the soil beneath the ship bow are where the soil is close to failing. This stress concentration may indicate the points that are undergoing a tensile stress as the particle are getting separated, or rolled over each other, leading to localized failure. From this point on, the ship starts to rise, while pulling the soil at the contact surface (see Figure 22), where the results could not be considered as realistic or reliable anymore and a lot of unrealistic stress oscillation could be seen caused by the ship pulling a certain area after each step.

According to the visualization of the final situation in Figure 26 and explanation above, despite all the adjustments and changes that have been made, as it can be clearly seen from the deformations, the results are not realistic in FEM. As it was predicted before the constant predefined contact area is considered as the main issue.

One possible solution for this issue is to perform the calculation in hundreds of different phases with each phase allowing the ship to move a couple of millimeters or centimeters. In each phase the contact area should be adjusted according to ship movement and soil deformation while the initial energy from one phase will be the final situation of the previous page. This solution of course is not a useful and feasible one due to the fact that the geometry of the problem has to be changed as well. The required time and attention to create the model is simply too much and unnecessary. Therefore, unfortunately similar problems are not suggested to be modelled using FEM.
4.1.2. **MPM Results**

Unlike FEM results, MPM results are much more convenient. In MPM the ship, after hitting the berm, transfers its energy into the soil. The energy is then being absorbed by deformations of the soil and lifting of the ship. It should be noted that due to limitation of current MPM version, the most ideal result in which the whole kinetic energy is being absorbed or transferred, has not been obtained. A lot of adjustments have been made to the model, by keeping the parameters the same, resulting in a number of different calculation models. The results shown in this part are from the model that had the most progress in terms of calculation steps (the most stable model).

Figure 27 shows the deformations at 6 different calculation time steps starting from 0.03 s (a) reaching 0.45 s (f). The colors are an indication for displacement of material points with the maximum displacement within the soil equal to 1.5 meters at f, while the total displacement magnitude of the ship is about 4.2 meters and reaching a velocity of around 8.4 m/s (10 m/s initially, Figure 28) after the impact before the calculation stops.

![Figure 27: MPM Ship-Berm Displacement](image)

![Figure 28: MPM Final Model Ship Velocity](image)
The deformations result is clearly following the expectation. As the ship is making contact with the berm, it starts pushing the soil and causing compression. Depending on the soil stiffness and internal friction angle (strength) the berm starts to force the ship to slide and be lifted (c and d). Due to high kinetic energy of the ship, the soil at the contact area, start to fail and loses all its strength. As a result the soil grains start to roll over each other. These particles will then act as an extra resistance in front of the ship (see Figure 27f). This phenomenon could only be included in MPM. Therefore, unlike FEM, the deformation and consequently the alternation of the ship and soil grains interaction is taken into account in the computation.

The shear stress changes in the model are important information to be obtained as it is expected to be the main failure mechanism. This was also the case in FEM model at early calculation stage. Figure 29 shows the increased shear stresses close to the ship bow at approximately 0.2 seconds after the impact. The coloring scale is from -50 kN/m² (dark blue) to 500 kN/m² (dark red).

![Figure 29: MPM Shear Stress Before Failure](image)

By looking at the result shortly after, approximately at 0.22 seconds, a failure plane becomes visible which is indicated with green arc in Figure 30.

![Figure 30: MPM Failure Plane](image)
Due to the increase in the shear stress, the soil between the ship and the green arc is being pushed away from the berm. In other words the soil is being detached and sliding on top of each other. After the first shear failure in the berm, the next failure happens at 0.3 seconds after the impact, see Figure 31.

Figure 32 visualizes the average shear stress in the soil in front of the ship bow. By looking at this graph it can be seen that the shear stresses are reaching to their maximum value right before the second failure is occurring where a substantial amount of soil is being pushed upwards by the ship. The sudden drop in shear stresses at 0.3 s, suggests the failure of the soil in this area leading to zero resistance.

Further, the horizontal stress changes have also been analyzed and are presented in Figure 33. The propagated horizontal stresses were important results, if the simulation could be fully performed. The calculation time steps in this figure are at the same exact time step of the deformation figure. The colors are considered only as an indication of stress difference within the berm. The issues with the output and visualization made it impossible to have a
perfectly presented result with a clear legend and stress range. The presence of three separated layer in berm model, which were necessary to obtain a uniform mesh, are responsible for the non-uniformity of stress situations at phases e and f.

![Images of stress propagation in MPM SHIP-BERM model](image)

**FIGURE 33: MPM SHIP-BERM HORIZONTAL STRESS**

Due to vibrations and oscillations of the material, it can be seen that the initial stress situation is not uniform from top to bottom; however this has negligible influence on the stress propagation after the ship impact. As the contact area is increasing and ship is pushing the soil to right as well as downwards, the stresses within the soil is increasing and propagating. The stress tends to have a more diagonal path rather than a horizontal path. The reason is while the ship starts to slide and tilt, more pressure will be available at the bottom side relatively to the upper part. Therefore the shearing of the soil particle leads to rotation of the main stress planes. Another factor is the failure of the soil in front of the ship. While the soil at top of the berm is failing it is then relaxed leading to a sudden decrease in stress. This is visible from the figure in phase e.

At steps e and f a change in stress situation is occurring. A part of it can be a result of rupture and failure in the soil. However, since the calculations have stopped, the reliability of the final steps is lower. Due to distortion of the mesh and floating material points, the material state values are not highly accurate leading to unrealistic results.

4.1.3. **FEM VS MPM**

Based on the explanations and visualization regarding the FEM and MPM results, the difference in both methods can be directly seen. While FEM has a lot of limitation for such a modelling, MPM could be considered as an ideal tool, while the issues at the moment have to be solved in future in order to have a stable calculation model.

Despite the differences and issues of both models, they both resulted in a similar failure mechanism in the soil which was discussed before and was expected. However the FEM
model is useful only for 0.2 second in this calculation, while the MPM model results follows the expectations throughout the whole simulation time.

As mentioned previously, an important advantage of MPM compared to FEM, is the fact that by any movement of the ship, the contact area is being adjusted automatically during the simulation. This provides the user to simulate a dynamic model with a more realistic result.

In addition, in MPM the ship is physically moving within the model which is a better representation of a real world problem. This is an important aspect since the initial impact force can highly influence the soil behavior. In FEM this situation is closer to just a pushing object rather than an initial impulse followed by a push. The sudden impulse at the impact can cause vibration and stress. This vibration may also lead to partial liquefaction of a sandy soil, decreasing its strength dramatically. Therefore this should not be neglected. The formulation of such a contact between two elements is not an easy task. It requires a great theoretical knowledge and understandings to be able to program such a calculation.

Furthermore, from the result it can be concluded that the MPM, as expected, is the better tool when it comes to large deformations even in static problems. In MPM the material points that store the material state and stresses can freely move through the mesh allowing for a better representation of a soil like material with grains.

![Figure 34: Deformation after approximately 0.5 seconds in both FEM and MPM](image)

### 4.2. Limitations

Both methods have their own limitation while modelling a dynamic model with a high kinetic energy. A common limitation for both methods is the buoyancy force in the presented ship-berm model. As mentioned previously, this force in reality is changing by any vertical displacement of the ship. More lifting means more ship weight resting on the berm. This increases vertical stress in the soil. In MPM, when the software is ready for commercial use in future, this issue can be tackled by running the model in two different simulations. Firstly apply a constant vertical force on the ship and when the results are obtained, back calculate the changes in buoyancy force by looking at vertical displacement of the ship then apply the findings in the model and recalculate.

Further, using the FEM for modelling a dynamic problem is highly limited. The objects are not separated therefore an initial impact of a moving object cannot be modelled. But when it comes to a highly simple 1D dynamic problem, as demonstrated in chapter 3, FEM becomes handy.
Chapter 4: Modelling results

In a non-symmetrical model, MPM could become very time consuming and using a lot of storage space. In the ship-berm model, the possibility of dividing the whole model into for example, two identical models, does not exist. On one side the ship is hitting the berm and is applying force and impulse. The stresses should then propagate and go through the whole length of the berm before reaching the main structure. Therefore the numerical model should whether be identical to the real sized problem or the whole model has to be downscaled. This should be taken under consideration when modelling similar problems.

Another MPM limitation which was encounter in this research was the maximum applied kinetic energy to the ship. A higher kinetic energy would apply a high impact force, collapsing the soil in front and causing instability in the computation. This issue is related to the contact formulation and programing and may only exist at current stage. In future and by further development of the software this might be solved.

4.3. Area of Application

All the created models in the previous sections have been made to simulate a real world problem, if not they can be adjusted and further improved to become applicable for similar simulations.

The static problem with a surface load is a simple and basis of a lot of more complex geotechnical simulations, such as consolidation or raft foundation calculations, which are already performed easily in engineering world. This model should be adjusted for any other application.

The simple dynamic model, however, can be directly recreated for similar impact problems. One possibility which is also related to ships and soils/berms is the impact of anchors being dropped on protective layers above an immersed tunnel or a sinking ship (Figure 35). Another similar example could be collision of a car on structures. These types of modellings in which no large deformations are expected and also having a constant contact area is sufficient, can be easily modelled with dynamic FEM that is similar to the presented 1D problem in previous chapter. In this model a simulation that provides both elastic and plastic deformation (MC model in Plaxis FEM) can also be used.

The result of MPM model indicates that when the software is ready, this exact problem can be modelled with realistic and reliable results. Further, all other dynamic problems that have similarity in any way with the ship-berm problem, can easily be adjusted and modelled; as an example, the analyses of a ship impact on a quay-wall at a harbor or sliding of the ship on top of the protection layer of an immersed tunnel in case of limited clearance.
CHAPTER 5:
SUGGESTED CALCULATION METHOD

As explained in previous chapter, the desired results are not obtained from the numerical modelling with both FEM and MPM and the fact that at the moment these tools are not usable for calculations, a calculation method is introduced and suggested. In this model a rough estimation of the propagated horizontal stress reaching the structure can be obtained.

5.1. SUGGESTED CALCULATION MODEL

Since the desired results have not been obtained due to limitation of numerical modelling at the moment, the combination of the analytical method and a numerical FEM model is suggested to be used for analysis of the ship-berm problem. The usage of FEM is due to the fact that for simple dynamic models it is a reliable tool, as verified before, easy to create and fully available at the moment. This suggested method is explained in this section and followed by a sample calculation.

The model uses an elastic material with no plastic deformation within the soil. The idea of having a fully elastic model is firstly the fact that this approach has been verified previously in this report; secondly a problem that includes a combination of plastic and elastic deformations, at an exact point, can be modelled as a linear elastic material with a different elastic stiffness value as demonstrated in Figure 36.

![Figure 36: Elasto-plastic vs Elastic](image)

By considering a fully elastic perfectly plastic material as a realistic situation, the exact point (indicated with a green dot in figure) can be reached following a simplified path. This basically represents the assumption in this suggested model.
In this model which combines the analytical and FEM, the minimum required berm length is obtained from the analytical method and the propagated stresses are estimated by recreating the ship-berm problem using a 1D dynamic FEM model. The relations in the FEM model follow the same principle as explained and verified in the dynamic model verification section which is a fully elastic 1 dimensional compression. Therefore the model has a width of 1 m (indicated with letter ‘a’ in Figure 37) where a blue square represents the ship with $V_m$ (ship velocity) showing the direction of the traveling ship and $\mu$ indicating the possible frictional resistance and its acting direction, see Figure 37.

![Figure 37: Suggested FEM Model](image)

The main goal for having this model is to keep the stresses the same so that the results can be directly used as definitive estimated values. The stresses reaching the structure at the end of the berm/model (left side of the shown model) are obtained using this model and are considered as the main outcome of these calculations and simulation. The method gives an indication of the maximum stress that will propagate and reach the protected structure.

Although the presented model may not be a perfect representative of the real world problem, especially with considering a linear elastic material, by adding layers on both sides of the model that only applies frictional resistance to the soil, the results can be improved to become more realistic. In other words, internal frictional resistance of the soil can be added to the model by creation of these two layers on top and bottom and allowing the soil to slide in between.

The process in order to get from the analytical calculation result to the FEM model is explained below.

**Step 1:** A number of parameters should be taken from the analytical calculations which are considered as input. These parameters are:

- The draft of the representative ship ($d$)
- The mass of the representative ship ($m$)
- The added water mass ($m_{\text{added}}$)
- The velocity of the representative ship ($V$)
- The calculated stoppage distance ($x$)
- The total length of the berm ($L$)

**Step 2:** Calculation of the initial stress based on the kinetic energy of the ship. With known values for velocity and total mass of the ship the kinetic energy is calculated using eq. 1. By considering that the kinetic energy is equal to the work done the initial force and stress can be calculated.

$$ W = F \cdot x \quad [kN \cdot m] \quad \text{(24)} $$
Assuming that the whole stoppage distance, taken from the analytical calculation, is the distance the force is being applied leading to the total work done, the force can be back calculated. This force is a constant force which is the average of initial force at the beginning and the final force which is considered to be zero. Using eq. 17 and assuming a constant contact area equal to the ship draft (d), the stress can be defined.

**Step 3:** Initialization and numerical modelling parameters determination. In step 3 the parameters for the FEM model are defined which are used as input. These parameters are:

- The dimensions of moving block (a) which has to be equal to 1 m.
- The length of soil model (L_m), which is n times smaller than the length of the berm (L). The value of n should preferably be kept as 1 since the propagated stresses are dependent on the total length.
  \[ L_m = \frac{L}{n} \quad [m] \]  
- The velocity of moving block (V_m).
  \[ V_m = V \quad \left[ \frac{m}{s} \right] \]  
- The stress in the model (σ_m). This value has to be equal to the stress calculated in step 2.
- The distance that the force is being applied by the block (x_m= x).
- The kinetic energy of the moving block (E_{kin,m}). This is again calculated based on the equality of the energy and work. By knowing σ_m and a, the force (F_m) is calculated which is then used to calculate the energy by being multiplied by x_m.
  \[ E_{kin,m} = W_m = F_m \cdot x_m = \sigma_m \cdot a \cdot x_m \quad [kJ] \]  
- The unit weight of the block (γ_m). In order to calculate the unit weight, firstly the mass of the block (m_m) is defined using the relation between mass, velocity and the kinetic energy.
  \[ m_m = \frac{2 \cdot E_{kin,m}}{V_m^2} \quad [kg] \]  
  By knowing the mass, the unit weight is given by:
  \[ \gamma_m = \frac{m_m \cdot g}{a^2} \quad \left[ \frac{kN}{m^3} \right] \]  
- The stiffness of the soil model (E_m). The soil stiffness is equal to the stress divided by strain where strain is calculated by dividing the deformation (equal to x_m) by the total length of the model (L_m). This relation is valid since the compression is only happening in 1 dimension and its direction is the same as stress direction (with a Poisson’s ratio equal to zero). Therefore no vertical stress and strain will exist in the model.
  \[ E_m = \frac{\sigma_m}{\varepsilon} = \frac{\sigma_m}{\frac{x_m}{L_m}} = \frac{\sigma_m \cdot L_m}{x_m} \quad \left[ \frac{kN}{m^2} \right] \]  

**Step 4:** By knowing all the necessary values the FEM model can be created.

A missing part in the given steps is the determination of the friction and interface of the two mentioned layers on top and bottom of the model. This is a part that requires more time and a detailed analysis. However if the internal friction angle of the modelled soil is being
considered as deterministic value in this case, by using eq. 31 [16][17] in Plaxis FEM the needed parameters can be defined.

\[
\tan(\varphi_{\text{soil}}) = R_{\text{inter}} \cdot \tan(\varphi_{\text{model}}) \quad [-]
\]

where:
- \( \varphi_{\text{soil}} \): Internal friction angle of soil \( [\degree] \)
- \( \varphi_{\text{model}} \): Chosen value for internal friction angle of the soil in model \( [\degree] \)
- \( R_{\text{inter}} \): Interface strength \([-]\)

In this model the whole mechanism that stops the ship is considered to calculate the elastic stiffness of the linear elastic model. However when it comes to stress propagation throughout the model having a fully elastic model means that no energy is being dissipated due to heat or plastic deformation. All the energy is being transferred into elastic model and stored as a potential energy. Therefore the stresses that reach the end of the model, where the structure is located, will be highly overestimated. The presence of the friction in the model created by adding two layers on top and bottom of the model is a tool to have a more accurate estimation for the propagated stress.

In case that the suggested model is being used a number of aspects should be considered:

- In real world situation the kinetic energy is being dissipated and transferred into heat (due to friction and plastic deformation) and potential energy (due to lifting of the ship and elastic deformation). By having a model which only considers elastic deformation, the whole kinetic energy is being stored in the model and transferred into potential elastic energy (similar to a spring); and since no energy is being transferred into heat and the fact that a linear elastic material is transferring all the stresses towards its base (the structure in this case). This result in an over estimation of the reaching stresses to the structure while in reality this stress is lower. Therefore it is important to include the suggested layers on top and bottom
- The existence of pore water pressure in the model is another point to be considered. The existence of water pressure, as an incompressible material, increases the total stiffness of the soil; however this also requires further analysis before application
- The stoppage distance of the modelled ship with FEM does not necessarily need to be equal to the one calculated using analytical solution. The analytical solution does not include soil parameters and is only being used for a rough estimation of the stoppage distance. Consequently in a model that the stiffness of the model and friction are taken into account, a comparison of the FEM and analytical solution is not logical

### 5.2. SAMPLE CALCULATION

In this section a sample calculation of the presented method in paragraph 5.1 is given. The calculation presents a real world problem including a bridge pier which is protected by a berm against ship collision. The used parameters for the ship (as input) are taken directly from an ongoing project. These parameters are as follow:

- Large Yacht
  - Length at waterline: 80 m
  - Width at waterline: 13 m
  - Draft: 4.5 m
  - Mass: 2500 tones
  - Added mass: 25% of total mass
  - Velocity: 15 knots (approximately 7.71 m/s)
Chapter 5:  
Suggested calculation method

- Sandy berm
  - Berm length: 50 m
  - Height: 9 m
  - Slope: 9.46 °
  - Internal friction angle: 30°
  - Friction coefficient: 0.36
  - Poisson’s ratio: 0

By using the given parameters above and the analytical stoppage distance method (presented in paragraph 2.2.2), the required needed length to stop the ship is estimated to be equal to **30 m**.

After defining the stoppage distance x (30 m) all the needed parameters in step 1 are known. Now all other needed values for creation of the Plaxis model can be calculated. This is done by following steps 2 and 3 explained in previous section. The final parameters are presented bellow while the detail calculations can be found in appendix D.

- The width of the block \( a = 1 \) m
- The length of the model \( L_m = 50 \) m
- Unit weight of the block \( \gamma_m = 6812 \) kN/m\(^3\)
- Velocity of the block \( V_m = 7.71 \) m/s
- The stiffness of soil layer \( E_m = 1147 \) kN/m\(^2\)

In the last step, step 4, the calculated parameters are used to create the Plaxis model. The created model is represented in Figure 38.

![FIGURE 38: SAMPLE CALCULATION PLAXIS MODEL](image)

As explained before all materials are considered to be linear elastic. The main parameters are defined in previous steps; therefore the dark blue square has a unit weight of 6812 kN/m\(^3\) with a stiffness of \(10^{10}\) kN/m\(^2\) and layers 1 to 3 with a stiffness of 1147 kN/m\(^2\), unit weight of 20 kN/m\(^3\) and zero Poisson’s ratio.

In order to allow the layer 1 to slide in between the other two layers interfaces have been added, indicated with red lines in the figure. The interface strength \(R_{inter}\) of the layers has been set to 0.01. This small value reduces the total interaction between soil layers. The virtual thicknesses of the interfaces are 0.1 m (the default value in Plaxis). At the red lines the vertical displacements are restricted while layer 1 can move horizontally. The green points are the chosen point for the result graphs where in point A the displacement will be checked and at point K the stresses. The values for stresses at point K are in the main outcome of this model which provides estimation for the maximum horizontal stress on the protected structure.

The model boundaries are shown with black lines and the movements in any directions are restricted.
The calculation procedure has been divided into two steps and performed twice. In the first simulation layers 2 and 3 are not activated while in second run, both layers and the interfaces are activated applying resistance to layer 1 through friction. The results are compared in Figure 39 and Figure 40.

**FIGURE 39: SUGGESTED MODEL DISPLACEMENT COMPARISON**

Figure 39 shows that the blue block in the calculation without layers 2 and 3 is compressing the soil more than the other simulation before bouncing back. The reduced displacement while the resistance are available at the sides and the fact that the maximum displacement is being reached in shorter time period, indicate that the kinetic energy of the block, apart from the elastic deformations of layer 1, is being absorbed/transfered by the availability of the side layers.

**FIGURE 40: SUGGESTED MODEL STRESS COMPARISON**
Chapter 5: Suggested calculation method

The same explanation can also be given on Figure 40. The stresses propagating towards point K is reduced due to the applied resistance of layers 2 and 3 suggesting that less kinetic energy is being directly transferred in elastic potential energy in layer 1. The difference in the stress oscillation between two calculations is increasing over the time indicating again that a certain amount of the energy is being lost due to friction after each calculation time step.

The amount of resistance and stress reduction in this model are influenced by the chosen properties of the interfaces and layers 2 and 3. These parameters are:

- The stiffness of layers
- The unit weight of layers
- Interface strength
- The dimensions of the side soils (increase in height results in increase in resistance)
- Material choice of the interfaces
- Virtual thickness of the interfaces

As mentioned previously and also can be seen from the high stress levels and displacement of the block, the simulation without the available frictional resistance on the side, provides overestimated values. Therefore it is essential to add the side layers. By including the layers on the side, as expected, the stresses are reduced however it cannot be considered as a highly reliable value since they have not been verified yet. Further research and a detailed analysis are required in order to be able to define the material properties for the side layers and interfaces that could be a better representative for a realistic situation.
CHAPTER 6: CLOSURE

This chapter will elaborate and evaluate the finding of this research. It also includes a suggestion for calculations and recommendation for future research.

6.1. CONCLUSIONS

This research analyzed the possibility of calculation of the protective berms using numerical models. The reliability of both Finite Element Method and Material Point Method for dynamic calculations was checked. An analytical method, defining a minimum stoppage distance of the ship, was also presented.

A comparison between the FEM, MPM final models and analytical was also expected in this research. The comparison could not be performed at a useful and required level due to the encountered limitations and issues, namely:

- The fact that FEM calculation is only usable for a fraction of seconds
- The MPM model also did not proceed enough through the simulation time
- The analytical method does not include the necessary soil parameters

However up to a certain extend the results can be discussed and compared. The discussions and conclusions are divided into five main categories that are discussed below.

Analytical method

The presented analytical method in chapter 2 provides estimation for the minimum required berm length in order to bring a ship to complete standstill. This method uses conservation of energy where the kinetic energy of the ship is transferred into potential energy and friction. The independency of the method on soil parameters makes this method not convenient in terms of Geo-engineering aspects. The surface friction between the ship and soil is the only parameter that is influenced by the berm material.

The stress propagations within the soil is also not mentioned and calculated in this method; as a result it does not provide any data on the stresses that will be applied to the protected structure in case of a collision. Therefore if knowing the stress on the structure is the main objective of any calculation, this method cannot be used. However despite all simplification within the model, it provides a rough estimation of the necessary required berm length. The calculation results are not highly accurate and the travel distance of the ship is overestimated since no deformation of the soil is considered. Nevertheless if no other tool is available, the analytical method remains an option and should not be totally neglected.
**Finite Element Method**

One of the objectives of this research was to create a FEM model that represents the ship collision on a berm. This was approached by firstly reliability checks and verifications for dynamic calculation using FEM. The results were satisfying and indicated the possibility and reliability of dynamic calculations within FEM. Therefore simple dynamic models can be easily modelled with FEM.

However when it came to the final model, it could be concluded that FEM lacks some ability to perform complex dynamic calculation. The ship and the berm had to be attached together in order to include any interaction in between, resulting in a constant contact area. Due to the high kinetic energy of the ship, having a constant contact area is a big issue. By any movement of the ship the contact area is continuously changing therefore more ship surface is being in contact with the soil resulting in an increased friction leading to higher resistance which could not be modelled in FEM.

On the other hand the ship-berm problem could not be approached by static modelling. The velocity and kinetic energy of the ship is the main factor in this model which cannot be included in a static calculation.

So the final outcome regarding the FEM ship-berm model is that such calculations are not suggested to be performed using FEM which results in a highly unrealistic results.

**Material Point Method**

Modelling using MPM was one of the main objectives of this research. The comparison of the simplified static and dynamic models between the verified FEM and MPM provided useful data. Despite the differences in both methods the observation concluded that MPM calculations are considered to be reliable.

Although due to limitation and the fact that MPM is still in development and has not reached its final stage, the MPM ship-berm model provided useful data, unlike FEM, allowing for a better analysis of the model. MPM clearly showed it capability for calculations of complex model, especially when it comes to dynamic problems. The movement of the ship after impact was highly realistic based on the set boundary conditions within the model. The soil deformation and propagation of the stress also followed the expected path.

The complexity of the formulation and the difficulty in creation of a stable model could be directly seen since several aspects could influence the computation or even cause it to crash.

The required calculation time is considered as a main disadvantage of this method; however the possibilities that provides, overcome this issue in complex dynamic calculations. The mesh size or number of element plays a critical role when it comes to calculation time and it should be considered and checked before running the simulation.

The MPM results suggest that, at this stage of the MPM development, the software is not yet stable enough to perform a simulation with availability of very high kinetic energy, as the ship-berm model. In this model the main objective was to estimate the stresses that will reach the protected structure which unfortunately could not be obtained since the simulations were not stable enough and did not proceed more than 0.5 seconds. Therefore at current stage this tool is not suitable for similar simulations and users should wait for further developments in the future.
Nevertheless by finalizing the MPM development in near future, this tool certainly has the capacity of performing complex and difficult simulations such as ship-berm model. It offers a lot of possibilities that are considered as a limitation in FEM especially in offshore applications. Therefore the development of different MPM formulations and approaches should be taken seriously as MPM is considered a main modelling tool for dynamic problems in near future.

**FEM vs MPM**

Through the performed verifications and the final model using both FEM and MPM a comparison between both methods could be made.

For simple static and dynamic methods, by comparing the obtained results, it can be concluded that the MPM calculation can be considered a reliable tool with a certain differences compared to FEM. In the simple static method both methods provided a failure load value very close to each other and the analytical calculation. The dynamic model results were also comparable while the MPM is clearly a more convenient tool for dynamic simulations where an element or object is moving through the model. This is because in MPM the elements can be created with a possible space in between while in FEM the elements have to be connected in order to have any interaction. This was evident from the block movement after the impact in the dynamic model where the block in FEM is constantly being pulled and pushed by elastic deformations of the soil (see Figure 19).

Regarding the main and final model it can be said that FEM is not a suitable tool for such simulations. It certainly lacks some crucial aspects and has some issues, namely the contact surface issue and the necessity of a physical connection between ship and berm for any interaction. But MPM, as explained above, showed its potential therefore it can be considered as a possible simulation tool for such a model, after the developments.

**General conclusions**

By performing the literature study and analysis, several other conclusions have been obtained that are not method specific.

Both FEM and MPM methods have their own limitation when it comes to very complex models. The difficulty in modelling a problem is not the only factor that makes a model complex, the interaction, deformation and stress situation are other aspects to be considered. The latter named factors affect the formulation and calculation procedure.

Since the soil resistance and reaction forces are highly dependent on the internal and external friction angle, the angularity of the soil grains has a high influence on the total resistance of the berm. The availability of pore water pressure and soil stiffness are other factors that influence the ship behavior in case of a collision.

**6.2. RECOMMENDATION FOR FUTURE RESEARCH**

For a better assessment and more reliable results regarding the ship-berm problem, the following recommendations are given:

- The lack of independency of the analytical method on geotechnical aspects should be investigated. A possible improvement on the calculations method and formulas would be required. This could be done by having a collected data from a collision or
performing scale model tests. Availability of a more accurate and realistic analytical solution allows for a better comparison with any numerical model

- By the time that the MPM software is ready and fully developed, a comparison of the numerical models with actual data or even the results of a test model is highly recommended. This will provide detailed information on the possible differences/limitations of MPM in similar problems

- Further improvement of the suggested method can be analyzed. Especially the influence of the frictional resistance on both side of the model, obtained by analysis of the influential parameters
BIBLIOGRAPHY


APPENDICES

APPENDIX A: FEM HORIZONTAL DYNAMIC INITIAL CHECKS

Appendix A presents the performed checks in order to transfer/rotate the verified vertical model to a horizontal model.

Restraining vertical movements
Gravitational force is the main aspect in this transformation. In vertical model this force is responsible for the gained velocity of the block while in horizontal model, the block has to be vertically fixed. Therefore a reaction force should be modelled. In Plaxis 2D, the resisting force can be applied using two methods: an equal upwards force uniformly distributed on the block or applying vertical restraints by prescribed displacement. These two methods are applied and compared with the results shown in Figure 41, for displacement and Figure 42 for vertical stress after 1 second of simulation. In both figures the reaction force is applied on the left block (indicated with upward arrows) and the vertical restraints on the middle block while the right block is free to move vertical.

![FIGURE 41: VERTICAL DISPLACEMENT CHECK](image_url)
By looking at the displacement result, it can be seen that the left block which has no restraints is being pulled 4.856 m downwards (1 second free fall) while the other two are being kept at their places. The stress situation is also showing a logical result for both left and middle block where due to restrictions the vertical stress is gradually and uniformly increasing from top to bottom. These stresses can be neglected and will not have any influence on the results if the block has a high stiffness and is fully rigid.

Since both two methods are showing the exact same result, it is easier and more convenient to use the vertical restrains using a prescribed line displacement instead of applying a line load; because the prescribed line displacement can be applied through the software interface regardless of any material property while the vertical loads have to be calculated based on the block mass.

**Applying horizontal kinetic energy**

In order to apply horizontal kinetic energy or a horizontal velocity, the same prescribed displacement option can be used. Instead of restricting any movements, the block can be displaced whether by applying constant or variable velocity, acceleration or even displacement. All three should give same result while following different path. The idea is to allow the block to reach a certain velocity after a defined time interval. So in case of using the displacement or acceleration option, the user has to calculate the required time for the block to reach the required velocity, while with the velocity option the time interval is not relevant.

These three options are applied all together in a same simulation and are compared. In order to reach the same values using all three options the following parameters have been used:

- Prescribed displacement= 4.905 m
- Prescribed velocity= 9.81 m/s
- Prescribed acceleration= 9.81 m/s²
- Calculation time= 1 s

The results are shown in Figure 43, Figure 44 and Figure 45 for displacement, velocity and stress respectively.
FIGURE 43: HORIZONTAL DISPLACEMENT CHECK

FIGURE 44: HORIZONTAL VELOCITY CHECK
In all three figures, slight differences within three methods are visible. By looking at the displacement, the block that is moving by an applied acceleration, is displacing less than the other two. This is due to the rounding of the values and is not considered as an issue. But when it comes to the velocity, the right block shows oscillation within the block. Although the average velocity is equal to the others but the differences between the right side and the left side is caused by the fact that the block is somehow being pushed from one side. The stress situation confirms this as well. In other words, in order to reach the required velocity, the block undergoes a sudden change in its stress situation and is pushed by a pulse like force.

This can be better described by looking at the Figure 46 where the displacement is shown after only 0.1 second.
The figure confirms that the left block is being pushed resulting in a sudden increase in the stress at the side where the impulse is being applied. Depending on the mass of the block or object this force is varying.

Although the stress oscillations and increments are negligible for a rigid block, it can be concluded that the best option to use is the prescribed velocity which does not require any other initial calculations and can be applied directly.
APPENDIX B: DYNAMIC FEM SENSITIVITY ANALYSIS

The sensitivity analysis has been performed on the FEM horizontal dynamic model. Three parameters that have the biggest influence on the results have been changed, namely: the soil stiffness, block mass and velocity. A higher value and a lower value of the initial model parameter are compared. The initial material parameters are: Soil stiffness \((E) = 50000\, \text{kN/m}^2\), Block mass \((m) = 10\, \text{tones}\), Block velocity \((V) = 20\, \text{m/s}\).

All three parameter changes result in expected changes. A higher stiffness allows less displacement while a higher kinetic energy (mass or velocity increase) results in higher displacement. Decreases in the named parameters show opposite results as expected.

Stiffness changes

![Displacement vs Time](image1)

**FIGURE 47: DISPLACEMENT VS TIME FOR STIFFNESS CHANGES**

![Velocity vs Time](image2)

**FIGURE 48: VELOCITY VS TIME FOR STIFFNESS CHANGES**
Mass changes

FIGURE 49: DISPLACEMENT VS TIME FOR MASS CHANGES

FIGURE 50: VELOCITY VS TIME FOR MASS CHANGES
Velocity changes

![Displacement vs Time](image1)

**FIGURE 51: DISPLACEMENT VS TIME FOR VELOCITY CHANGES**

![Velocity vs Time](image2)

**FIGURE 52: VELOCITY VS TIME FOR VELOCITY CHANGES**
APPENDIX C: DYNAMIC FEM VS MPM

Appendix C provides the result for the comparison between the FEM and MPM. The comparison has been done for three set of parameters. In all three datasets similar results have been obtained with a conclusion that MPM can be considered as a reliable tool for dynamic calculations. The comparison is discussed in detail in section 3.1.4.

Dataset 1
- Block unit weight ($\gamma$) = 10 kN/m$^3$
- Block velocity ($V$) = 20 m/s
- Soil stiffness ($E$) = 10000 kN/m$^2$

![Soil Displacement vs Time](image1)

**FIGURE 53: FEM AND MPM SOIL DISPLACEMENT VS TIME, DATASET 1**

![Block Velocity vs Time](image2)

**FIGURE 54: FEM AND MPM BLOCK VELOCITY VS TIME, DATASET 1**
Dataset 2
- Block unit weight ($\gamma$) = 20 kN/m$^3$
- Block velocity ($V$) = 20 m/s
- Soil stiffness ($E$) = 10000 kN/m$^2$

\[\text{FIGURE 55: FEM AND MPM SOIL DISPLACEMENT VS TIME, DATASET 2}\]

\[\text{FIGURE 56: FEM AND MPM BLOCK VELOCITY VS TIME, DATASET 2}\]
**Dataset 3**
- Block unit weight ($\gamma$) = 30 kN/m$^3$
- Block velocity ($V$) = 20 m/s
- Soil stiffness ($E$) = 10000 kN/m$^2$

**FIGURE 57**: FEM AND MPM SOIL DISPLACEMENT VS TIME, DATASET 2

**FIGURE 58**: FEM AND MPM BLOCK VELOCITY VS TIME, DATASET 3
Appendices

APPENDIX D: STEPS 2 AND 3 OF THE CALCULATION SAMPLE

Appendix D provides the detail calculation of steps 2 and 3 for the sample calculation presented in paragraph 5.2.

Step 2:

\[ E_{\text{kin}} = 0.5 \cdot (m + m_{\text{add}}) \cdot V_{\text{ship}}^2 \] [kJ]

\[ E_{\text{kin}} = 0.5 \cdot 2500 \cdot (1 + 0.25) \cdot 7.71^2 = 92881.41 \text{ kJ} \]

\[ F = \frac{W}{x} = \frac{E_{\text{kin}}}{x} \] [kN]

\[ F = \frac{92881.41}{30} = 3096.05 \text{ kN} \]

\[ \sigma = \frac{F}{d} \] [kN/m^2]

\[ \sigma = \frac{3096.05}{4.5} = 688 \text{ kN/m}^2 \]

Step 3:

\[ L_m = \frac{L}{n} \] [m]

\[ L_m = \frac{50}{1} = 50 \text{ m} \]

\[ V_m = \frac{V}{s} \] [m/s]

\[ V_m = 7.71 \text{ m/s} \]

\[ \sigma_m = \sigma \] [kN/m^2]

\[ \sigma_m = 688 \text{ kN/m}^2 \]

\[ x_m = x \] [m]

\[ x_m = 30 \text{ m} \]

\[ E_{\text{kin,m}} = W_m = F_m \cdot x_m = \sigma_m \cdot a \cdot x_m \] [kJ]

\[ E_{\text{kin,m}} = 688 \cdot 1 \cdot 30 = 20640 \text{ kJ} \]

\[ m_m = \frac{2 \cdot E_{\text{kin,m}}}{V_m^2} \] [kg]

\[ m_m = \frac{2 \cdot 20640}{7.71^2} = 694 \text{ tonnes} \]
\[
\gamma_m = \frac{m_m \cdot g}{a^2} \left( \frac{kN}{m^3} \right)
\]

\[
\gamma_m = \frac{694 \cdot 9.81}{1^2} = 6812 \left( \frac{kN}{m^3} \right)
\]

\[
E_m = \frac{\sigma_m}{\varepsilon} = \frac{x_m}{L_m} = \frac{\sigma_m \cdot L_m}{x_m} \left( \frac{kN}{m^2} \right)
\]

\[
E_m = \frac{688 \cdot 50}{30} = 1147 \left( \frac{kN}{m^2} \right)
\]