Final Project Report

Submitted By: Arman Moussavi, PhD Student

Dept. of Civil & Environmental Engineering

Northwestern University

Date Submitted: 12/09/2022

Contents

Static Compressive Loading of a Hollow Pipe

Arman Moussavi

ABSTRACT: The purpose of this report is to model buckling and deformation caused by a compressive load applied to a pile (pile and pipe will be terms used interchangeably throughout this report). This will be done by simulating a hollow, Structural ASTM-A36 steel, cylindric pile with static compressive loading on the top face and fixed support on the bottom face. It is expected that this system will buckle when loading is applied if the material properties are not suitable for the predicted loading. This model will be studied through finite element analysis methods using Abaqus. Abaqus is a software developed to work with finite element analysis and computer-aided engineering. This report works to understand how an insufficiently designed pile will react to compressive loading. The insufficiently designed pile is expected to deform due to compression, possibly showing buckling behavior. Analyzing the stress contours of this system reveals the regions that are most prone to deformation in a loading situation. Additionally, the critical buckling load is estimated to be roughly 9,125 psi. The deformation of the pile is determined as the displacement of the non-fixed end of the pile ("top" plate). The displacement of the "top" plate at buckling/failure is determined to be roughly 1.4 inches. These values are determined through multiple tests applying various loads to the developed model. Overall, this report provides a finite element analysis of the buckling/failure of a hollow cylindrical pile using Abaqus.

KEYWORDS: *Finite Element Analysis, Abaqus, Pipe, Buckling*

INTRODUCTION

When constructing various types of structures, for example, a bridge, it is necessary to design pilings that can withstand the axial force of the maximum predicted load. In the case of a bridge, pilings must be designed to be able to withstand extremely large axial compression without deforming. If the piles are not properly designed, the bridge may have an unpredictable failure causing a large catastrophe. It is important for the designing engineers to have a deep understanding of the material properties of the piles they will work with, as well as the predicted loads that must be resisted. A miscalculated resisting load onto piles without proper material properties could cause the piles to compress and deform. If piles supporting a structure such as a bridge were to deform, it could cause uneven load distribution and potential failure of the entire system. It is important to design the piles to the proper material property and strength requirements to avoid catastrophic failures in the future. This report will analyze a hollow steel cylinder that may represent a system such as a shallow foundation pile. It is important to note that while the dimensions of this pile may not be the most practically accurate, the purpose of this report is to show the behavior of the materials and pile geometry in buckling failure. However, if promising findings are discovered, new methods of pile geometry and design may be warranted as future research interests.
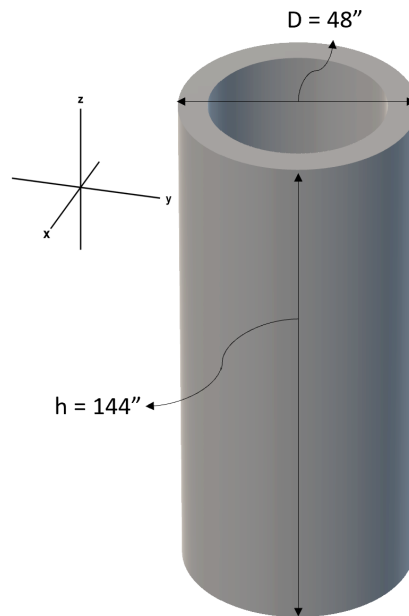
METHODOLOGY

Model layout

Fig. 1. Hollow cylindrical pile model.

Figure 1 provides an image of a hollow cylindrical pile that will be tested throughout this report. The piling is modeled similar to that of a pipe. The length of the pipe is modeled at 144 inches or 12 feet long. The total diameter of the pipe is set as 48 inches or 4 feet wide.
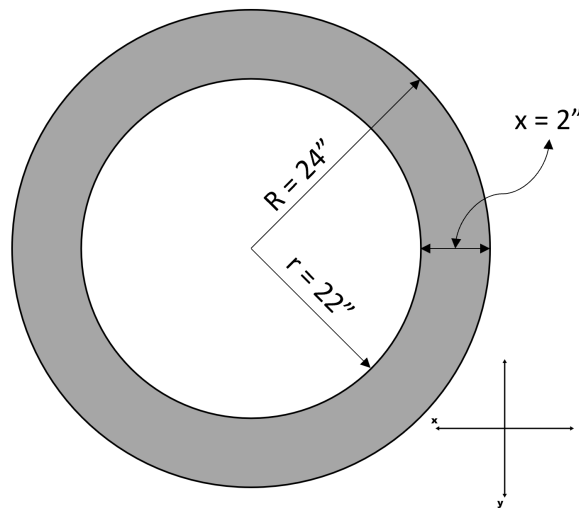

Fig. 2. Hollow cylindrical pile cross-section.

Figure 2 provides the cross-section of the hollow cylindrical pile model. It is seen that the outer radius "R" is 24 inches or 2 ft. The inner radius "r" is 22 inches, or 1 foot 10 inches. This concludes the thickness of the pile is 2 inches. Table 1 displays the material properties used in the modeling of this pile obtained from "Grade guide: A36 steel 2021".

Table 1. Material Properties for Steel, Structural ASTM-A36

Modulus of Elasticity	Poisson's Ratio	Mass Density	Friction Coefficient
29,000 ksi	0.32	0.28 lb/in^3	0.25

Model Construction

A model must be created using the tools of Abaqus in order to run finite element analysis using the software. The general system layout and construction will be outlined in this portion of the report. However, detailed instructions on the steps to replicate the model will not be listed (if interested, please contact the PI of this report for the extended model set-up).

To model the identified problem, there are three necessary components. One cylindrical pipe following the specified dimensions of Figure 1 and Figure 2 as well as 2 fixed rigid plates that will provide Abaqus a surface for interaction forces and boundaries. Figure 3 shows a part created for the pipe that will be analyzed in this report.
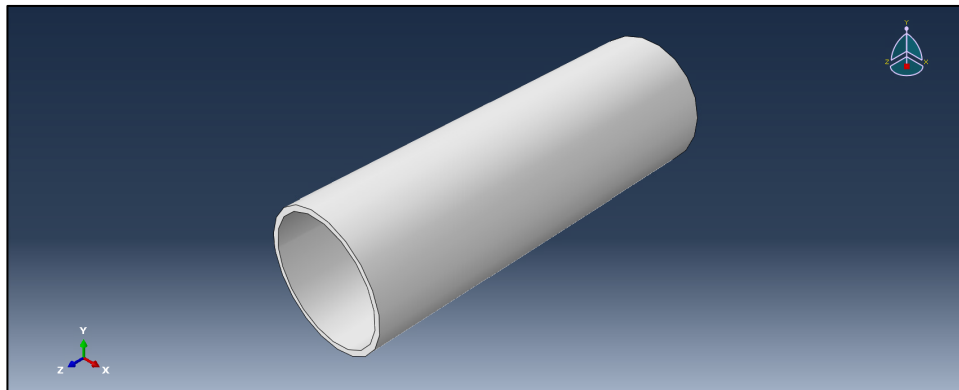

Fig. 3. Part: Pipe

Figure 4 shows a rigid plate that is created for later use in interaction in order to apply loads on the pipe as well as set boundary conditions.
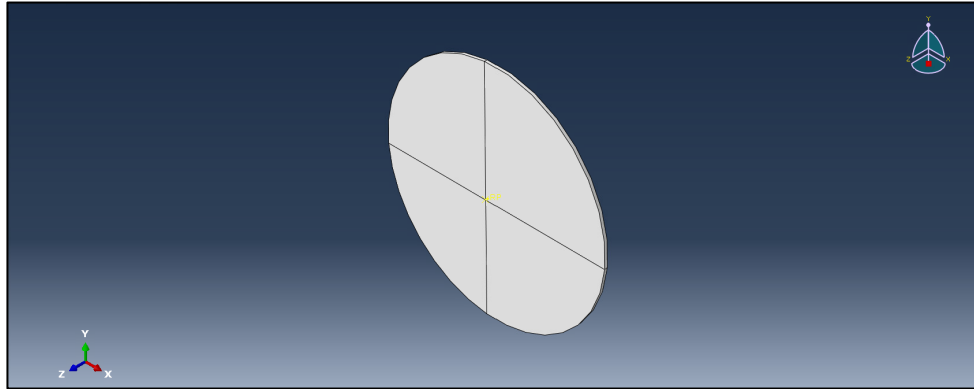
Fig. 4. Part: Rigid Plate

When assembling the pipe to the rigid plate, the translation tool can be utilized in Abaqus's "Assembly" module to replicate the rigid plate on both ends of the pipe as seen in Figure 5.
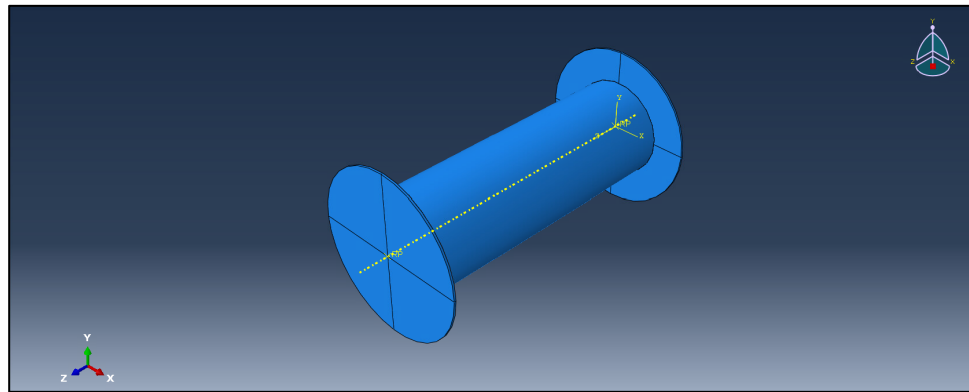

Fig. 5. Assemblage of parts

Next a static general step is created using the default settings. Then, the interactions between the rigid plates and the deformable pipe are set. This is done using the "Create Interaction" and "Create Interaction Properties" tools. Boundary conditions must then be set for the rigid plates. The "bottom" plate is taken as a fixed boundary using the Symmetry/Antisymmetry/Encastre boundary condition type where the selection "ENCASTRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0)" is used. The "top" plate is set using the Displacement/Rotation boundary condition type where all directions except U3 (corresponding to the z-axis) are set as 0 (fixed). The loading is applied to the "top" plate as a pressure with uniform distribution. The magnitude of this pressure is varied as seen later throughout the analysis of this model. Figure 6 displays the bounded and loaded model.
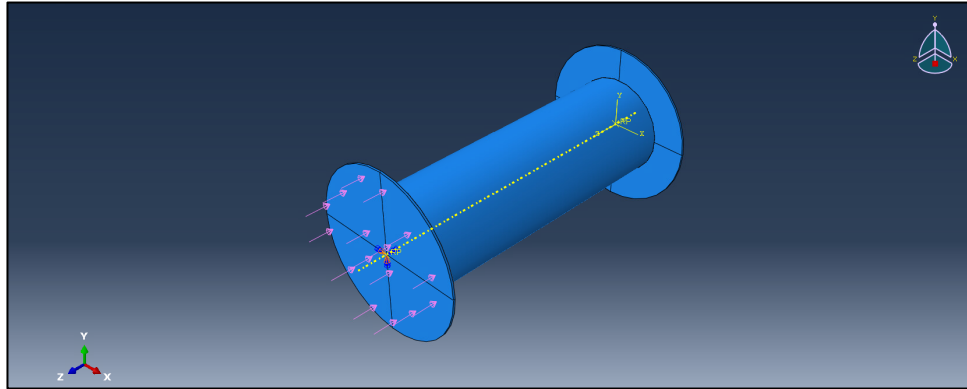
ME327

Fig. 6. Model with boundary conditions and load applied

Next the parts must be individually seeded. The rigid plate is seeded at an approximate global size of 3. The pipe is also seeded at an approximate global size of 3. Figure 7 shows the seeded and meshed assembled model.
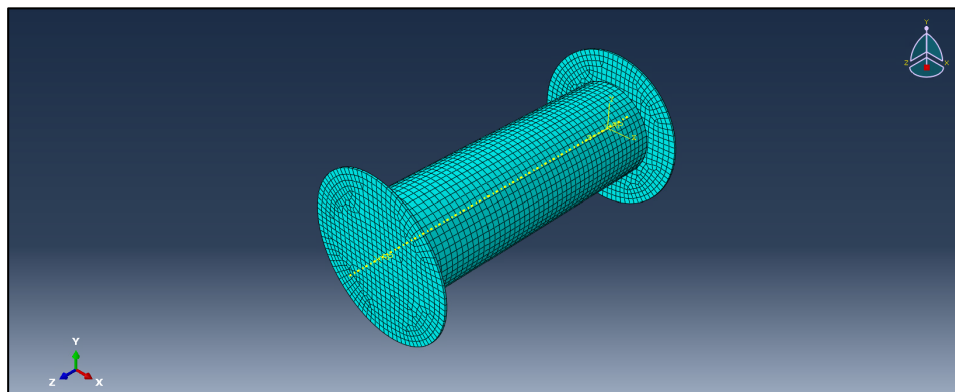

Fig. 7. Model with seeding on all parts

Finally, the model is ready to be analyzed by Abaqus. It is necessary to create a job and submit that job to obtain finite element analysis results. The results obtained for this report will be presented and analyzed in the next section.

RESULTS AND DISCUSSION

The first analysis conducted for this pile is the estimation of critical buckling load. This estimation was done by running the created model with various applied pressure loads and analyzing the results. A range of 8,000 – 10,000 pounds per square inch is tested for the pile. When visualizing the deformed models, buckling can be seen at loads starting from 9,130 psi and continuing to the

tested 10,000 psi. Figure 8 displays a stress contour plotted over the deformed shape of the pile subjected to 10,000 psi. The main failure region is identified by the black circle outline. This region will be the focus of a stress analysis conducted in the S33 (third component of the principal stress) direction specified by Abaqus (corresponding to the z-direction).
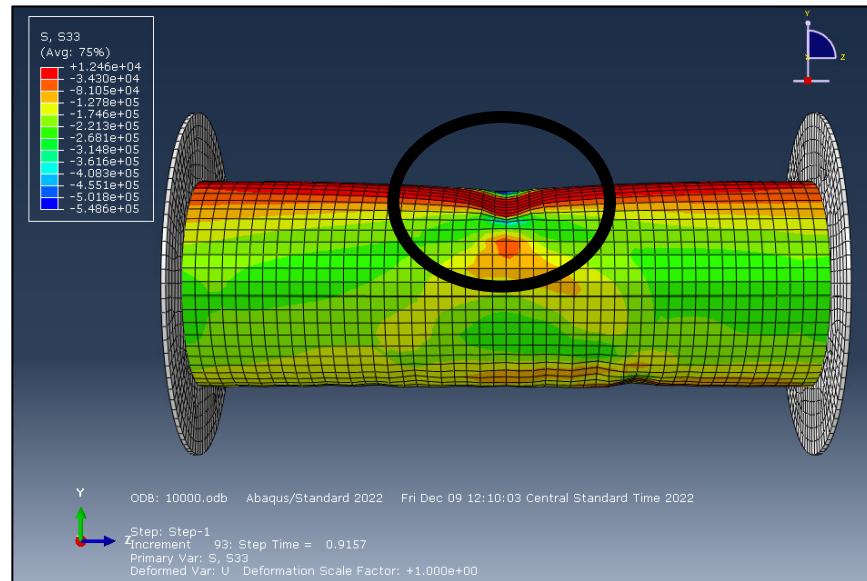

Fig. 8. Deformation and failure due to 10,000 psi application

To analyze this region, a path is created along the x-y plane as seen in Figure 9. This path will allow an analysis of stresses at the buckling region of the pile.
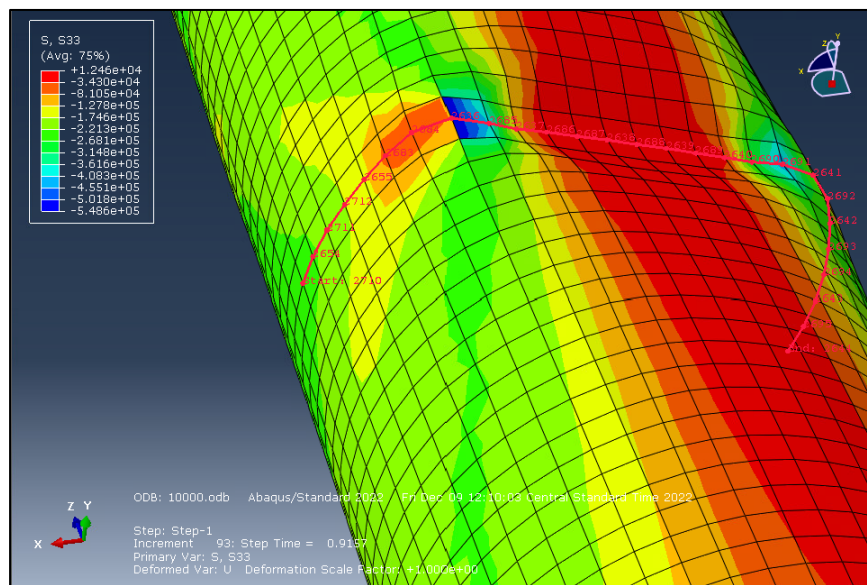

Fig. 9. Path specification on 10,000 psi application

Table 2 shows the maximum third component of the principal stress determined along this analysis path using the true distance for x-values for each loading model conducted. Figure 10 is a graphical representation of Table 2, provided to easily see the physical difference in values obtained.

Table 2. Maximum S33 (third component of the principal stress) in the region of buckling (specified path Fig. 9)

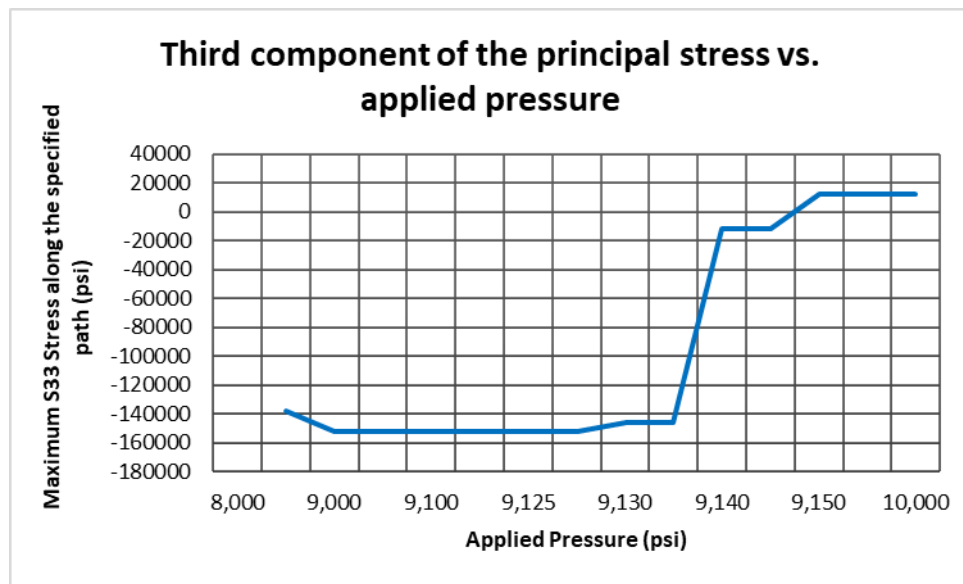| Applied Pressure (psi)	8,000	9,000	9,100	9,125	9,130	9,140	9,150	10,000
Maximum S33 Stress along the specified path (psi)	-137827	-152648	-152444	-152362	-146583	-11432.3	12099.5	12099.5

Fig. 10. Maximum S33 stress evaluated for varying applied pressure

It is seen that running trials with loads from 8,000 – 10,000 psi give a large spectrum of stress values in the third component of the principal stress direction. At lower loads, the stress is in a constant range around -150000 psi. However, when the load transitions from around 9,130 – 9,140 psi, it can be seen that there is a very large jump in stress. This is found to be correlated with the buckling of the pile visualized in Figure 8. Because of this information, it can be determined that this pile will show buckling behavior/failure once the applied pressure reaches roughly 9,125 psi.

Additionally, the deformation of the pile is analyzed. This is done by evaluating the U33 displacement (z-direction) of the "top" rigid plate that holds the pressure load applied. Table 3 shows the displacement of the "top" rigid plate for each pressure loading trial. Figure 11 is a graphical representation of Table 3, provided to easily see the physical difference in values obtained.

Table 3. U33 displacement (z-direction) of the "top" rigid plate

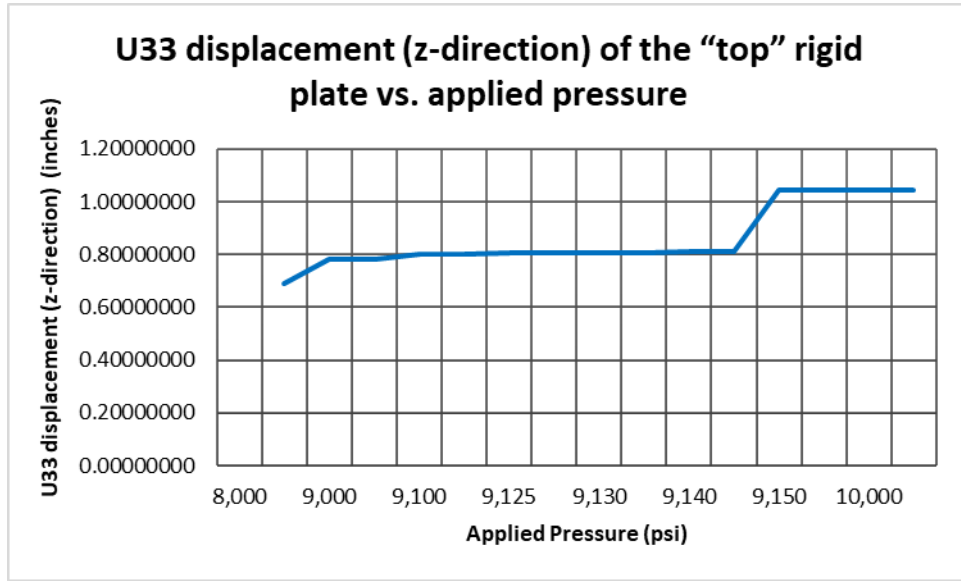| Applied Pressure (psi)	8,000	9,000	9,100	9,125	9,130	9,140	9,150	10,000
U33 displacement (z-direction) (inches)	0.691187	0.783947	0.79968	0.804662	0.806883	0.811202	1.04198	1.04198

Fig. 11. U33 displacement (z-direction) of the "top" rigid plate for varying applied pressure

At lower loads, the displacement is in a constant range around 0.8 inches. However, when the load transitions from around 9,140 – 9,150 psi, there is a distinct large jump in displacement. This is found to be correlated very closely with the increase in the third component of the principal stress at the buckling/failure of the pile Figure 10. Because of this information, it can be determined that the buckling/failure of the pile will cause a displacement of the "top" plate by roughly 1.4 inches.

CONCLUSIONS

This project demonstrated the utilization of Abaqus to compute finite element solutions. Using programs such as Abaqus has the potential to drastically decrease the computation time of finite element methods and help visualize problems. Additionally, Abaqus has the potential of omitting simple algebraic mistakes throughout the computation.

In this report, Abaqus was used to analyze the stress and deformation of a hollow, Structural ASTM-A36 steel, cylindric pile. It was determined that the pile modeled here will show buckling

behavior/failure once the applied pressure reaches roughly 9,125 psi. It was determined that buckling/failure of the pile will cause a displacement of the "top" plate by roughly 1.4 inches. Additionally, the displacement of the "top" plate is found to be directly related to the increase in the third component of the principal stress at the buckling/failure of the pile. These are extremely important values to note for engineering practices such as structural and foundation design. When determining what materials and dimensions are needed for a pile design, it is important to know what loads will cause the pile to fail and in this case buckle. Additionally, the displacement of whatever structure may be supported by the pile heavily depends on the buckling of the pile. For example, in this case, if a structure that this modeled pile is supporting causes over 9,000 psi of pressure on the individual pile, there is a major risk of buckling/failure. If the pile were to buckle, the displacement of the support to the structure may cause uneven load distribution and support to the structure, inducing factors of torsion, stress, as well as instability, which can lead to significant damage or even collapse. This can be particularly dangerous if the structure is occupied at the time. In addition to the potential for physical damage, displacement of foundation piles can also cause cosmetic issues in buildings, such as cracks in the walls or floors. In conclusion, it is crucial to consider these factors in the design process to ensure the safety and integrity of structures.

RECOMMENDED READING

1. Abaqus Analysis User's Guide, Abaqus 6.14. http://130.149.89.49:2080/v6.14/
2. Abaqus Analysis User's Manual (6.12). http://dsk-016-1.fsid.cvut.cz:2080/v6.12/books/usb/default.htm?startat=pt04ch11s03aus67.html
3. "Finite element analysis of buried steel pipelines under strike-slip fault displacements": https://www.sciencedirect.com/science/article/pii/S0267726110001442

REFERENCES

[1] Grade guide: A36 steel. Metal Supermarkets. (2021, March 25). Retrieved December 9, 2022,

from https://www.metalsupermarkets.com/grade-guide-a36-steel/