Final Project Report

Submitted By: Arman Moussavi, PhD Student

Dept. of Civil & Environmental Engineering

Northwestern University

Date Submitted: 03/15/2023

Please note that this report is constructed for LEARNING PURPOSES ONLY as an assignment for

2023_MECH_ENG_426-1_SEC20_AND_CIV_ENV_426-1_SEC20

ME_426

Contents

ABSTRACT 3

KEYWORDS 3

INTRODUCTION 3

METHODOLOGY 4

Model layout 4

Model Construction 5

RESULTS AND DISCUSSION 8

CONCLUSIONS 13

RECOMMENDED READING 14

REFERENCES 14

Ballistic Impact Endurance of Stainless-Steel AISI 450

Arman Moussavi

ABSTRACT: The purpose of this report is to understand the failure mechanisms of steel plates under ballistic impact. The motivation of this project is the increasing awareness of firearms in the United States and their devastating consequences on the community. This is done by modeling a 9 mm bullet projectile fired at a stainless-steel AISI 450 square plate. The model is constructed and studied through finite element analysis methods using Abaqus. Abaqus is a software developed to work with finite element analysis and computer-aided engineering. The bullet is modeled as a rigid body for the sake of simplicity as well as the focus on the plate response. The stainless-steel plate has fixed supports on all edges to prevent rotation and translations. The bullet is given velocity in only the direction normal to the plate to allow for impact. For various thicknesses of the stainless-steel plate, differing failure behavior is observed. Plugging due to penetration of the projectile was observed and determined to have a high dependency on the initial thickness of the plate. Additionally, the dissipation of strain energy after impact holds dependency on the thickness of the plate. No solid conclusions are drawn for the Von Mises stress of the system throughout the simulation, however, there appears to be some dependency on plate thickness. Overall, this report provides a finite element analysis of a stainless-steel plate under ballistic impact using Abaqus and provides a warrant for the further understanding of the failure mechanics and properties of ballistic impacts.

KEYWORDS: *Finite Element Analysis, Abaqus, Ballistic, Stainless-steel AISI 450*

INTRODUCTION

A very prominent and continuously growing issue of gun violence has plagued the United States for many years, becoming a topic of extreme debate and discussion. In efforts to combat this terrible issue in today's society, the study of material response to ballistic damage is of important interest. This study specifically investigates the mechanical response of stainless-steel AISI 450. Stainless steel may be used in the construction industry for a variety of purposes such as roofing, cladding, and structural elements due to its strength, durability, and resistance to corrosion. In the tragic scenario of a bullet passing through a building wall, understating how a lining of stainless steel would respond may motivate more gun-violence-conscious construction and designs. While it is not always expected that material will prevent certain projectile punctures, it is interesting to map out the damaged regions that may make the material more prone to failure in following incidents. This report aims to provide insights into the response and failure mechanisms of a stainless-steel plate subjected to a ballistic impact scenario involving a common bullet, by analyzing the plate's response at various thicknesses.

METHODOLOGY

Model layout

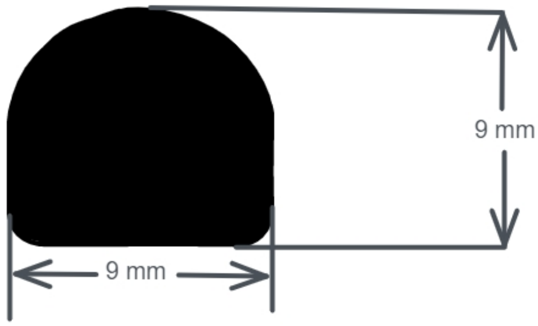


Fig. 1. 9 mm bullet projectile.

Figure 1 provides the dimensions of a 9 mm bullet projectile that will be utilized throughout this report. The specimen decision was based off the heavy presence of the ammunition, as the 9 mm is one of the most popular handgun rounds in the world. This projectile will be modeled as a rigid shell insusceptible to deformation. This assumption is made for simplicity of the model to focus efforts on the analysis of the plate response.

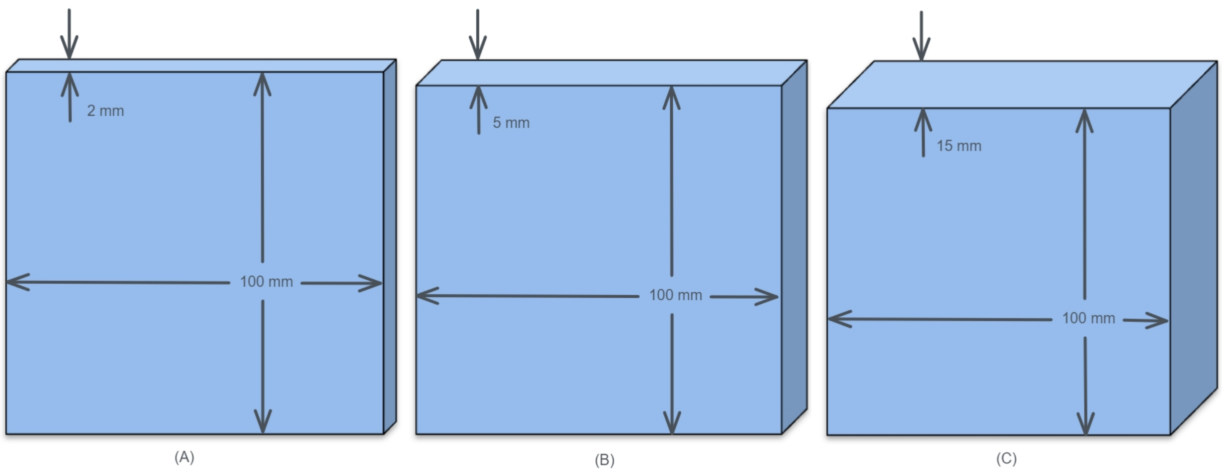


Fig. 2. Stainless-steel plate with thickness of (A) 2 mm (B) 5 mm (C) 15 mm.

Figure 2 provides the dimensions of each plate tested in this study. The length and height are held constant at 100 mm each. The thicknesses examined are 2 mm, 5 mm, and 15 mm. The plates will be modeled as stainless-steel AISI 450. Table 1 displays the material properties used in the modeling of this plate obtained from Mansur et al.

Table 1. Material Properties of stainless-steel AISI 450

Modulus of Elasticity	**Poisson's Ratio**	**Density**	**Yield strength**
210 GPa	0.3	7800 kg/m^3	814 MPa

Model Construction

A model is 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 projectile following the specified dimensions of Figure 1 as well three plates of varying thickness as seen in Figure 2. Figure 3 shows a part created for the projectile that will be analyzed in this report.

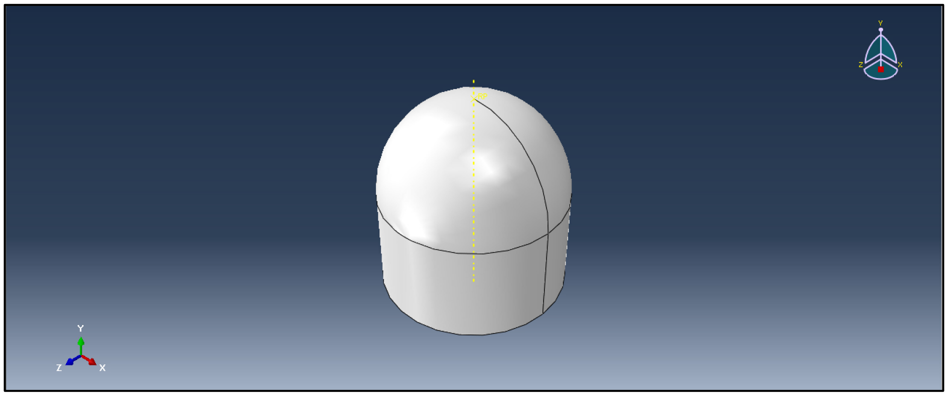


Fig. 3. Part: Projectile

Figure 4 shows a 2 mm thick plate that is created for ballistic impact testing. A circular partition is made in the center to redefine a more accurate mesh scenario for testing.

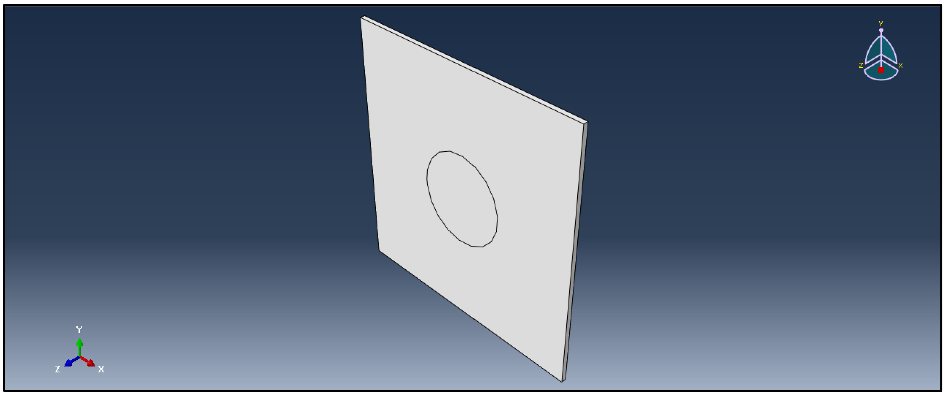


Fig. 4. Part: Plate

When assembling the bullet and plate, the rotation and translation tools may be utilized in Abaqus's "Assembly" module to indicate a proper impact direction scenario as seen in Figure 5.

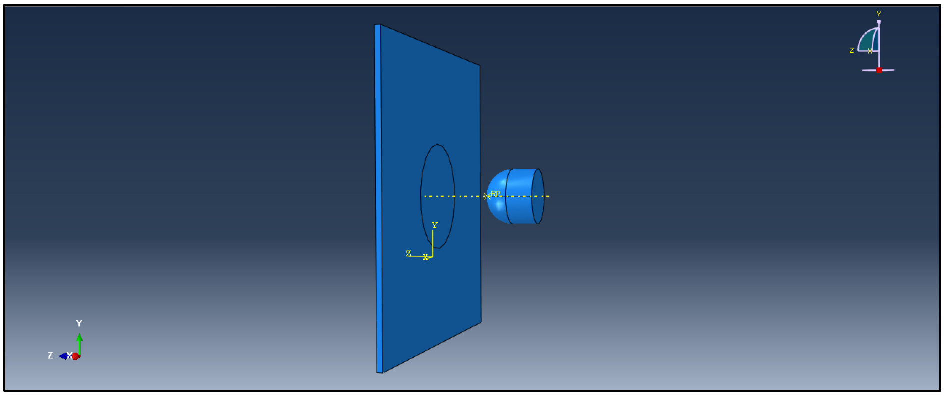
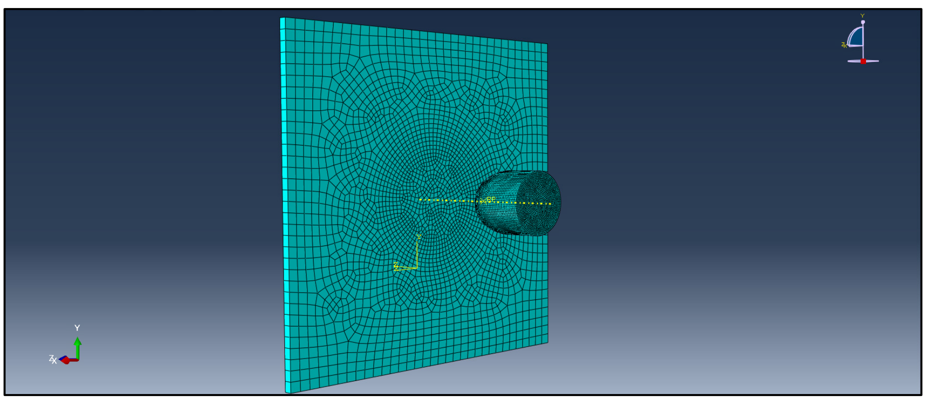
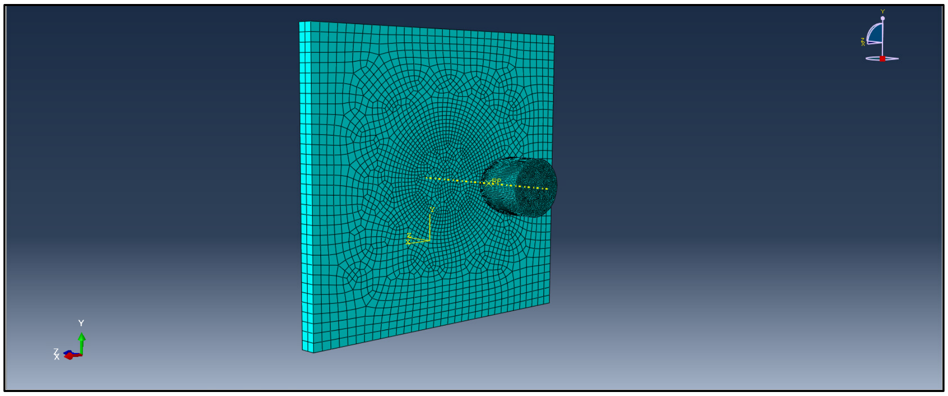


Fig. 5. Assemblage of parts

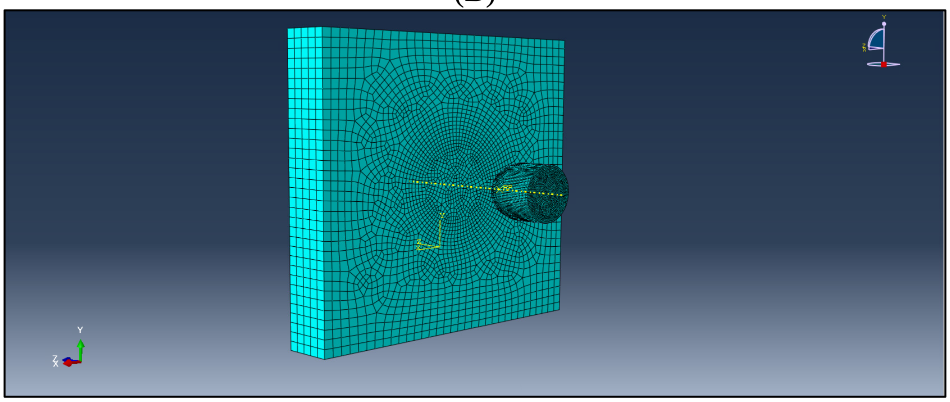
Material properties are assigned to the plate parts only to allow for deformation and analysis using the parameters of Table 1 (Mansur et al.). Next a dynamic, explicit step is created at a time period of 0.008. The stable increment estimator is set to global and improved dt methods is assigned along with nonlinear geometry. Then the model is meshed. The bullet is meshed at 5588 elements, however this is assumed to be arbitrary as the bullet is a rigid body. The plates are meshed at 2620, 5234 and 13085 elements for 2 mm, 5 mm, and 15 mm thickness respectively. The circular partition in the middle of the plate may be seeded separately to contain a higher concentration of elements as it is the main area subject to deformation. Figure 6 shows the meshed assembly for each simulation set-up.



(A)



(B)



(C)

Fig. 6. Meshed assembly of thickness: (A) 2 mm (B) 5 mm (C) 15 mm

Next, load specifications are made to the model. For the plates, ENCASTRE conditions are set for each side face to prevent the displacement or rotation of the perimeters. A reference point is assigned to the tip of the cone of the bullet. Then the reference point is assigned a velocity of 370 m/s in the “33” direction and zero in all other directions. The velocity of the projectile is determined from average 9 mm-caliber round handgun firings (Mitchell, 2020). Figure 7 displays the prescribed boundary conditions.

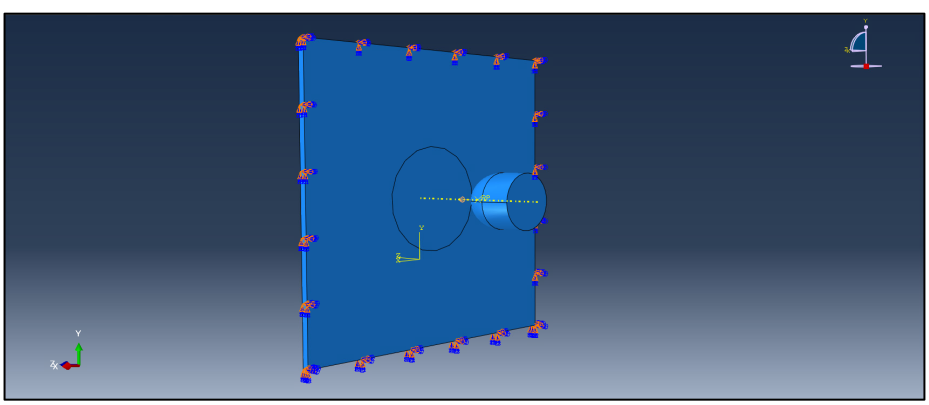


Fig. 7. Boundary conditions

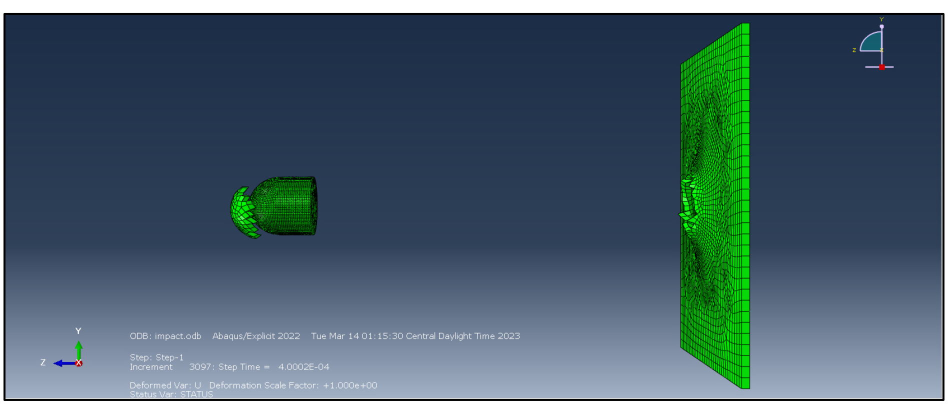
Next interactions are defined for the model. A contact interaction is assigned with tangential and normal behavior properties. The penalty method is used in friction formulation with an assigned isotropic friction coefficient of 0.2.

Finally, a job is created for the simulation and submitted for analysis. This process is then repeated identically with the exception of plate thickness. Once all three models are successfully created and simulated, finite element analysis results are available for analysis. The results obtained for this report will be presented and analyzed in the proceeding section.

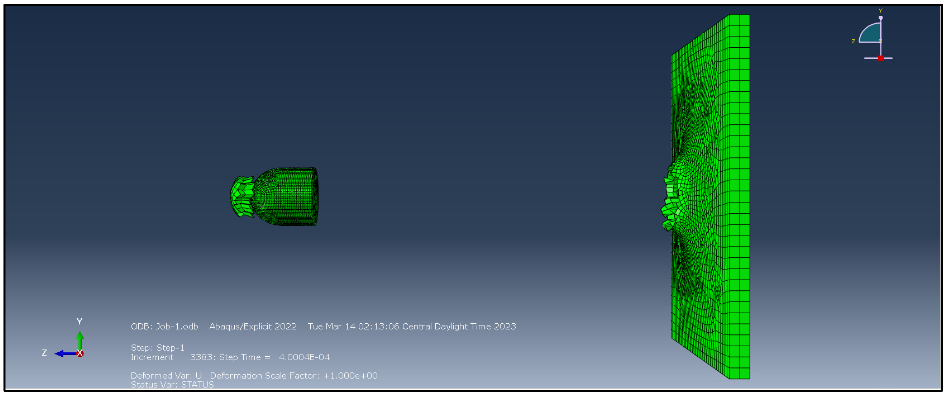
RESULTS AND DISCUSSION

The first analysis conducted for this ballistic impact analysis is the deformation of the plate. This analysis is based on qualitative observations from the visualization of each model.

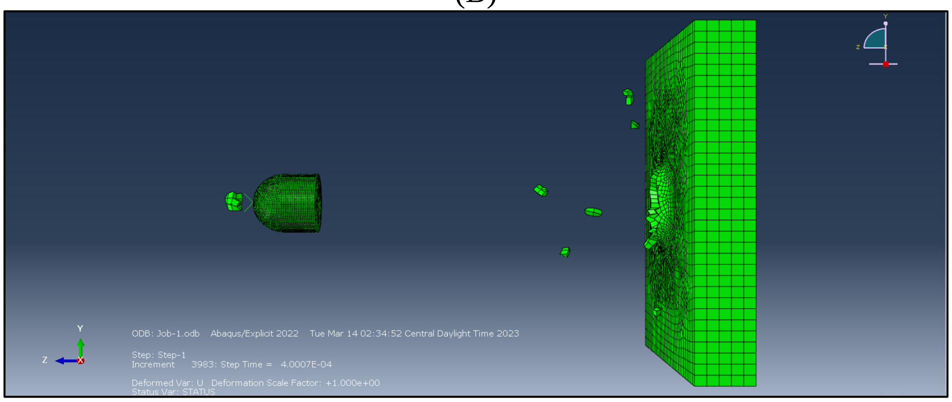
Figure 8 displays each model after the first simulation step is taken. In each sample, obvious failure is shown from a puncture hole due to the path of the bullet. It is also interesting to note the varying degrees of plugging for each sample. There is a clear trend correlating plug size to initial thickness of the plate. It is seen that a thinner plate Fig. 8 (A) allows for wide plug, nearly the size of the bullet cross-section, to be displaced from the plate. In Fig. 8 (B) there is an obviously thicker plug but smaller in size compared to the cross-section of the bullet. Fig. 8 (C) displays the smallest plug in comparison to the cross-section of the bullet. However, another interesting observation is made for the 15 mm thick plate. It seems that the thickness of the plug is not representative of the thickness of the plate (i.e. the plug's thickness is not obviously close to the thickness of the plate). This is due to the mode in which the plate failed. Instead of an even fracture/punch through, the elements broke apart and flew in different directions. This is assumed to be related to the relation of the material properties to the thickness of the plate.



(A)



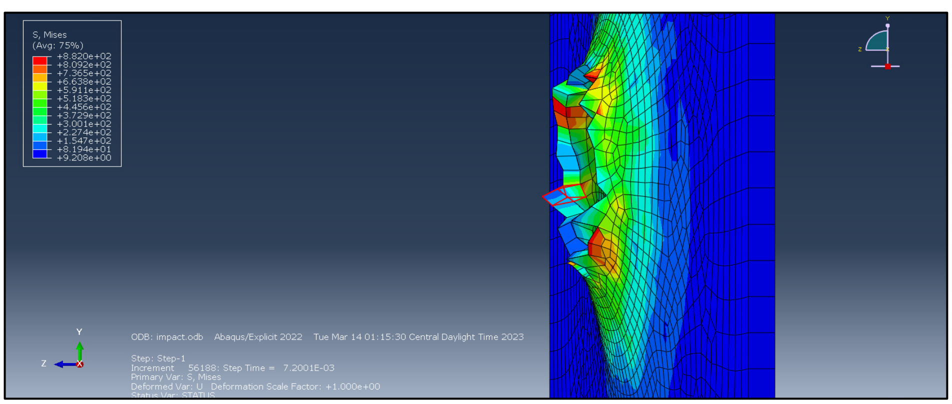
(B)



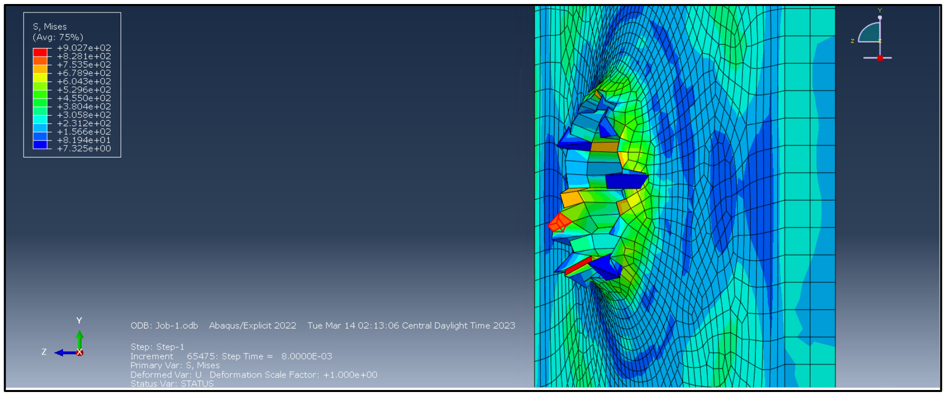
(C)

Fig. 8. Simulated deformation of thickness: (A) 2 mm (B) 5 mm (C) 15 mm

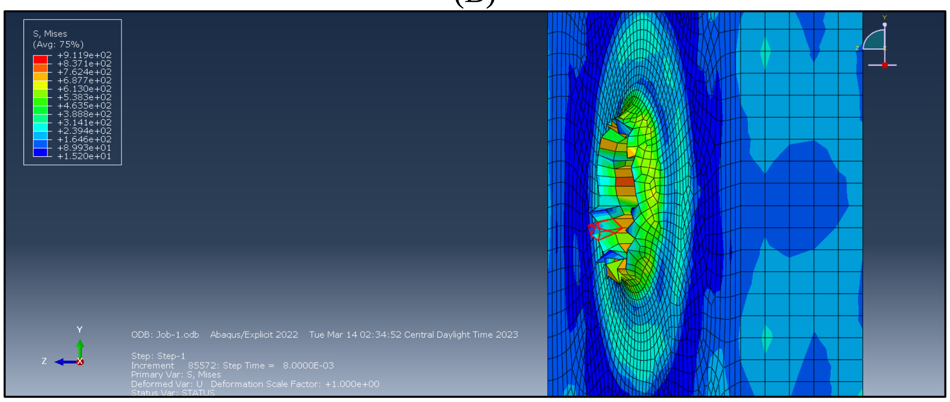
Next, the simulation is analyzed at the final step. Figure 9 presents each plate and its fracture at the end of the simulation. Additionally, some arbitrary element is selected along the perimeter of the fracture (indicated by red outline). This is done to extract useful information about the strain energy and stress.



(A)



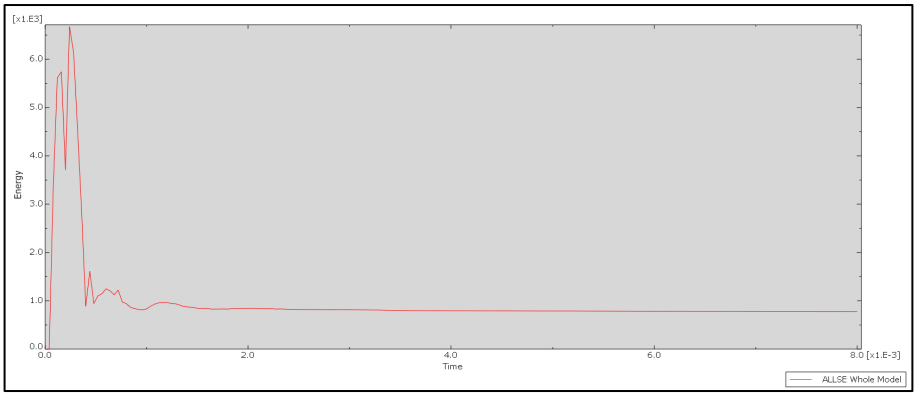
(B)



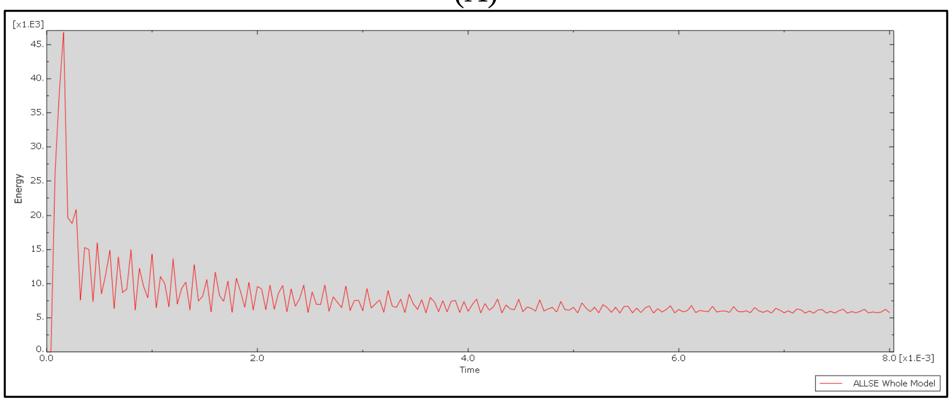
(C)

Fig. 9. Simulated deformation of thickness: (A) 2 mm (B) 5 mm (C) 15 mm

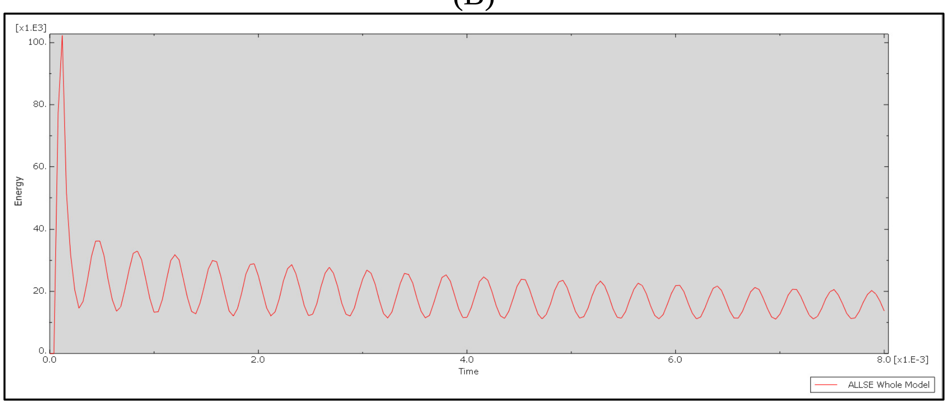
Figure 10 displays the plot of strain energy for each element specified in Fig. 9. There is a very similar and apparent trend seen in each plot where energy hits a maximum, then dissipates out. The maximum is cause by the fracture of the plate, when the bullet completely breaks through. The maximum of each plot is tabulated in Table 2. The general trend of strain energy oscillations as dissipation is observed. It is noticed that as the thickness of the plate increases, there is an up-scaling of the strain energy of elements on the fracture surface. The 2 mm plate shows very rigid peaks and values, possibly due to the fast reflection of the impact wave through the plate. The energy is dissipated very quickly to a constant value. The strain energy dissipation of the 5 mm plate is represented by slower dissipation of the initial impact wave. However, the dissipation still causes the strain energy to approach a constant value. The 15 mm plate shows a much smoother function of strain energy dissipation. Additionally, the oscillations of strain energy caused by the impact wave is not seen to fully dissipate within the experimental time frame. However, there is a decline observed and it is expected to return to a constant value at longer allowed time frames. These results confirm a large thickness dependency of materials on their ability to dissipate energy.



(A)



(B)



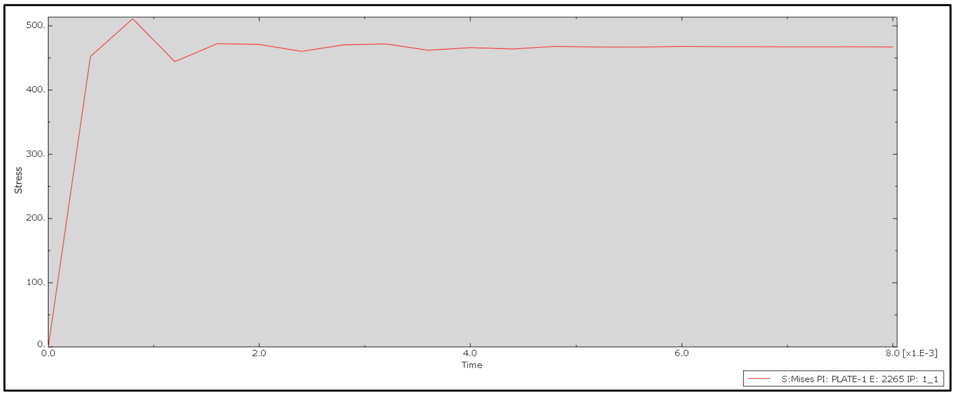
(C)

Fig. 10. Strain Energy of arbitrary element along fracture as a function of time;
Plate thickness: (A) 2 mm (B) 5 mm (C) 15 mm

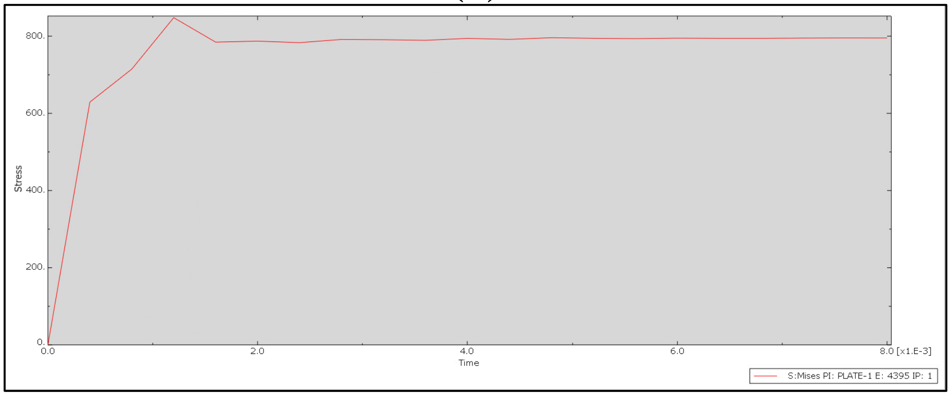
Table 2. Maximum Strain Energy

Plate Thickness:	2 mm	5 mm	15 mm
Maximum (simulated) Strain Energy	6,678.74	46,836.2	102,284

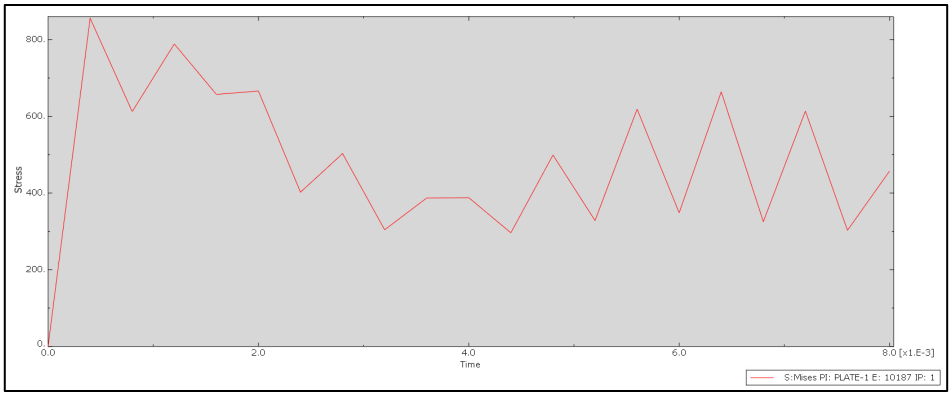
Figure 11 displays the plot of Von Mises for each element specified in Fig. 9. These results show very interesting trends. The maximum Von Mises stress is tabulated in Table 3. There seems to be some stress up-scaling when increasing the plate thickness, however it is not nearly as straight forward as the strain energy dissipation. The maximums are seen again near the time of fracture/penetration of the bullet however there is a much greater variance of simulation time at which these maximums occur. It is also interesting to note the convergence of stress for the 2 mm and 5 mm plate. It is assumed that this converged value of stress is due to plastic deformation as well as the convergence of dissipating oscillations seen in strain energy. However, the 15 mm plate shows very different results. After reaching a maximum, the stress decreases but then oscillates heavily throughout the remainder of the simulation. A potential explanation of this behavior may be due to the strain energy, that is not yet dissipated at the end of the simulation as observed previously, causing coordinating stress waves throughout the plate, leading to the observed variations. From these results, it is concluded that there does seem to be a dependence of Von Mises stress of the simulation on plate thickness, however, the mechanism that governs this dependence is not well understood. This warrants further investigation into the reasoning for the behavior observed in this report.



(A)



(B)



(C)

Fig. 11. Von Mises stress of arbitrary element along fracture as a function of time;
Plate thickness: (A) 2 mm (B) 5 mm (C) 15 mm

Table 3. Maximum Von Mises Stress

Plate Thickness:	2 mm	5 mm	15 mm
Maximum (simulated) Von Mises Stress	510.967	848.01	855.881

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 deformation, strain energy, and Von Mises stress of three stainless-steel AISI 450 plates with thicknesses of 2 mm, 5 mm, and 15 mm subjected to a ballistic impact of a rigid 9 mm bullet. Trends in plugging due to penetration of the projectile are seen to have a high dependency on the initial thickness of the plate. The dissipation of strain energy after impact is also seen to have an obvious dependency on the thickness of the plate. Although not tested in this report, it is assumed that both of these results also will hold high dependency on the medium/material parameters of the plate, producing correlated results to what is observed here. Finally, there are observed differences in the Von Mises stress after impact depending on plate thickness, however, no impactful conclusions are drawn.

Overall, this paper warrants further investigation into the effect of material property as well as the characterization of stress dependency in ballistic impact testing. It is important to understand how the properties of different systems have impacts on their behavior in general. In this scenario, proper characterization of failure mechanics will result in material optimization based on the desired results. While this is a very oversimplified as well as a broad goal, it is representative of an overarching goal of material/structure design and development for the future.

Please contact the author at ArmanMoussavi2027@u.northwestern.edu for any supplementary material/information not provided

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. Kumar, M., Deep, U., & Dixt, P. M. (2017). Simulation and analysis of ballistic impact using Continuum Damage Mechanics (CDM) model. Procedia Engineering, 173, 190–197. https://doi.org/10.1016/j.proeng.2016.12.057

REFERENCES

Mansur, A., Nganbe, M. Assessment of Three Finite Element Approaches for Modeling the Ballistic Impact Failure of Metal Plates. J. of Materi Eng and Perform 24, 1322–1331 (2015). https://doi.org/10.1007/s11665-015-1406-4

Mitchell, G. (2020, April 22). How far can a bullet fired from a handgun travel? BBC Science Focus Magazine. Retrieved March 14, 2023, from https://www.sciencefocus.com/science/how-far-can-a-bullet-fired-from-a-handgun-travel/