



James Burton

Application of Computer Aided Engineering Processes

University of Wales Trinity Saint David
ACDB5007 - Computer Aided Engineering

Ordnance, Munitions and Explosives (BEng)
Academic Year: 2025 - 2026

Supervisor: Andrew Noble

Abstract

This report is to demonstrate the critical analysis and evaluation techniques for structural problems and Computer Aided Engineering (CAE) results, including a deep understanding of the application of software packages to investigate engineering structural problems and the use of cognitive skills in the formulation and analytical solution of an explicit linear model of a projectile being fired at a plate.

This analysis includes understanding of the benefits and limitations of the software used in the case study, as well as all implicit functions and variables used by the software to create the most accurate model simulation possible. Validation of the model simulation will be represented from the results in the form of deformation, velocity, normal stress and elastic strain comparison between materials and pre-existing literature, where modelling has been used in conjunction with real world testing.

A mesh independence study has also been conducted to assess the impact of mesh sizing and the accuracy of the results, with a point of mesh independence clearly outlined in the report.

The influence of boundary conditions has been analysed to assess its influence on how they may affect the outcome of a simulation.

This report also includes a discussion of CAE tools used in industry, where they may excel or where there may exist limitations for that software in real world applications.

Keywords:

Ansys, Modelling, Simulation, Mesh, Analytical Software, Case Study.

Table of Contents

Abstract	ii
List of Figures	iv
List of Tables	v
List of Abbreviations	v
1 Introduction.....	1
2 Case Study.....	2
2.1 Bullet and Plate.....	2
2.1.1 Modelling.....	2
2.1.2 Coordinate Systems	6
2.1.3 Mesh Generation.....	6
2.1.4 Initial Conditions	8
2.1.5 Analysis Settings	9
2.1.6 Results	10
2.1.7 Mesh Independence Study.....	22
2.1.8 Boundary Condition Influence	26
2.2 Validation	27
2.2.1 Validation of Aluminium Plate.....	28
2.2.2 Validation of Copper Plate.....	29
3 Analytical and Software Comparison.....	32
3.1 Ansys Workbench	32
3.2 Abaqus.....	32
3.3 Fusion 360	33
4 Conclusion.....	34
5 References	36

List of Figures

Figure 1 - List of Explicit Dynamics function in Ansys Workbench	2
Figure 2 - Model of Bullet and Plate	3
Figure 3 - Ansys Workbench Library Dashboard	4
Figure 4 - Imported Materials list	4
Figure 5 - Material Selection for Plate	5
Figure 6 - Coordinate System Creation	6
Figure 7 - Sphere of Influence	7
Figure 8 - Meshed Model Including Sphere of Influence	8
Figure 9 - Edge Fixture as Part of Initial Conditions	9
Figure 10 - Solution Progress Monitor	10
Figure 11 - Initial Impact of Projectile on Aluminium Plate	11
Figure 12 - Initial Impact of Projectile on Copper Plate	11
Figure 13 - First Penetration of Aluminium Plate	12
Figure 14 - First Penetration of Copper Plate	13
Figure 15 - Final State of Aluminium Plate	14
Figure 16 - Final State of Copper Plate	14
Figure 17 - Deformation Graph of Aluminium	15
Figure 18 - Deformation Graph of Copper	16
Figure 19 - Elastic Strain in Aluminium.....	17
Figure 20 - Elastic Strain in Copper.....	17
Figure 21 - Elastic Strain Graph for Aluminium	18
Figure 22 - Elastic Strain Graph for Copper	18
Figure 23 - Stress in Aluminium	19
Figure 24 - Stress in Copper	20
Figure 25 - Stress Graph for Aluminium	20
Figure 26 - Stress Graph for Copper	21
Figure 27 - Velocity Graph of Projectile for Impact on Aluminium plate.....	21
Figure 28 - Velocity Graph of Projectile for Impact on Copper Plate	22

Figure 29 - Sphere of Influence Reduction and Mesh Element Size Increase .	23
Figure 30 - Non-Penetration of Mesh Due To Element Size	23
Figure 31 - Mesh Convergence Point.....	24
Figure 32 - Mesh Independence Point	25
Figure 33 - Change of Fixture of Plate	26
Figure 34 - Fixed Face Result	27
Figure 35 - " <i>Bullet impact and penetration of target plate at normal direction at various velocities</i> ". Adapted from [8].....	28
Figure 36 - " <i>Sheet metal sample penetrated by a 9mm Luger bullet with full metal jacket of standard velocity, see shot No. 3 and 4 from impact side (a) and output side (b)</i> ". Adapted from [11].....	29
Figure 37 - " <i>Typical paint detachment patterns (post-impact): (a) radial cracks propagated to a maximum radius for 2 mm thick copper plates</i> ". Adapted from [12].	30

List of Tables

Table 1 – Comparison Table of Properties of Aluminium	28
---	----

List of Abbreviations

CAD	Computer Aided Design
CAE	Computer Aided Engineering
CAM	Computer Aided Manufacture
FEA	Finite Element Analysis

1 Introduction

Computer aided design (CAD), computer aided manufacturing (CAM) and computer aided engineering (CAE) are powerful software tools used by engineers globally to design, model and simulate complex engineering problems in a controlled environment in order to understand how something might operate. This ranges from simple concept validation to complete safety analysis, incorporating a range of effects such as structural, chemical and thermal effects when a model is subjected to a load. This allows designers and engineers to theoretically proof a model against its intended purpose in the safest, most efficient way possible.

Whilst computer software was developed in the mid-1940's for complex automated calculations to compute radar related vectors, the first coining of the term CAD was by that of Douglas T. Ross, where he and his team identified that it was an efficient and viable way to create simple circuit diagrams [1].

With regards to CAE, the term itself was coined by Dr. Jason A. 'Jack' Lemon, a founder of Structural Dynamics Research Corp., in the late 1970's [2]. However, this isn't to say that CAE wasn't utilised before then, with the earliest recorded example of commercial CAE/CAM being PRONTO, first developed in 1957 by Dr Patrick Hanratty. This specifically paved the way for modern day explicit solvers used in Ansys. [3].

Shortly after in 1963, Ivan Sutherland developed the first graphical user interface and light pen used for technical drawing, allowing engineers to easily design objects and annotate digital drawings [4].

In the present day, CAE has developed into the ability not only to program, but also simulate events, with industry leaders such as the previously mentioned Ansys, as well as Dassault Systèmes, and Siemens offering cutting edge software services, with branches of each tailored for specific use.

2 Case Study

As part of this assignment, A simple linear motion impact simulation has been set up, in the form of a 9mm bullet and 2mm thick plate, by using Ansys workbench.

2.1 Bullet and Plate

The bullet and plate model represents a basic linear motion impact at high speeds, where variables such as materials, velocity, mesh sizing and any fixture of parts can affect the outcome of the simulation.

2.1.1 Modelling

The model for the bullet and plate itself was provided as part of a premade template, and imported into workbench using the 'Geometry' function under 'Explicit Dynamics', as seen below:

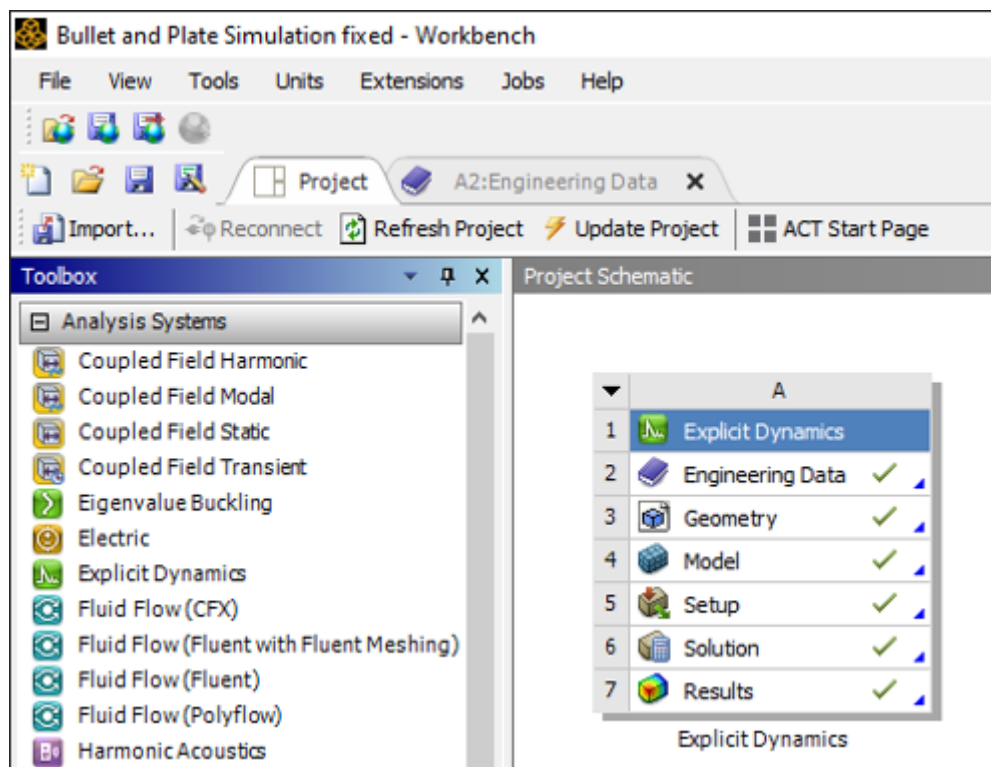


Figure 1 - List of Explicit Dynamics function in Ansys Workbench

It was important to set up the geometry prior to setting up the engineering data, to allow the software to understand what to apply the materials to.

The geometry of the bullet and plate can be seen below:

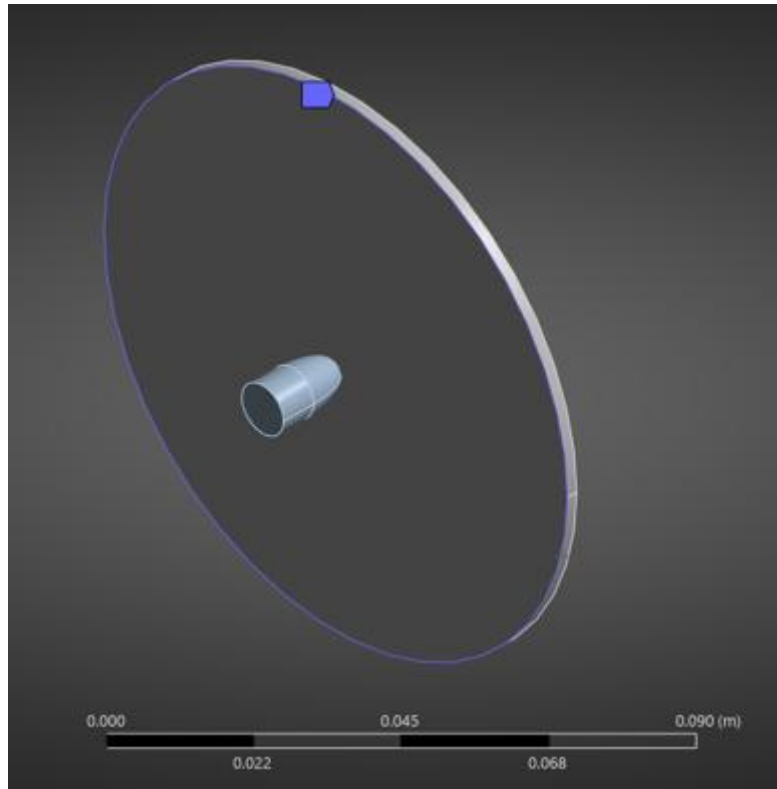


Figure 2 - Model of Bullet and Plate

Following this, materials were imported using the 'Engineering Data' function. One of the benefits of Ansys is that it contains a library of pre-saved material properties, however properties can be imported externally from Granta or other material libraries. If these fall short of the material required then Ansys also allows the creation and addition of new materials into its library, as well as the adjustment of existing material properties should it prove that they do not contain the requirement information for the explicit solver.

The library of materials can be seen in figure 3.

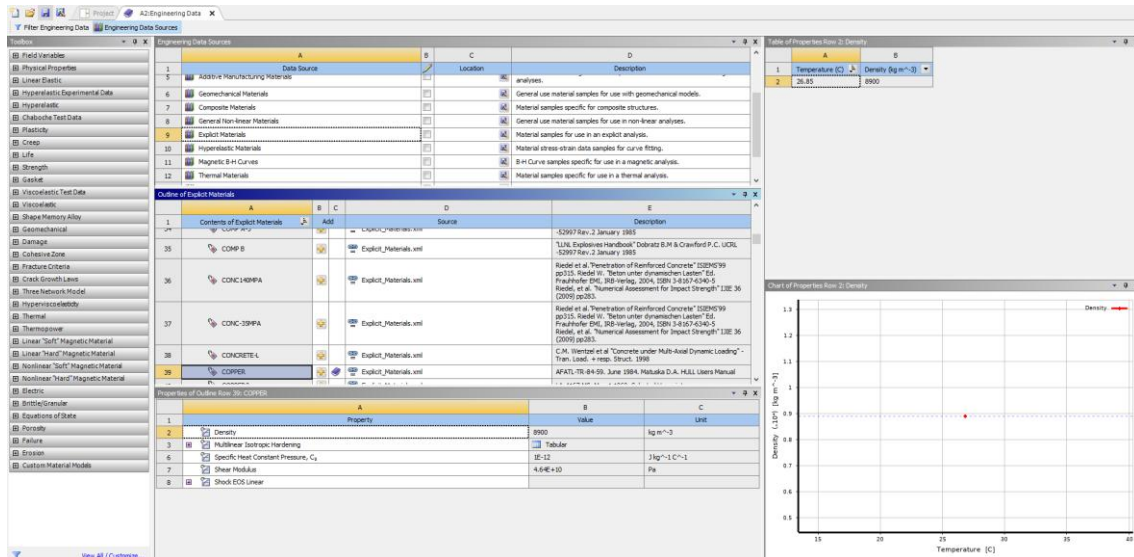


Figure 3 - Ansys Workbench Library Dashboard

For this simulation, the materials used were lead antimony for the bullet, with a generic copper and AL7039 aluminium being used for different solutions for the plate.

A dashboard of the materials imported into the project, along with the material properties for copper, can be seen in figure 4.

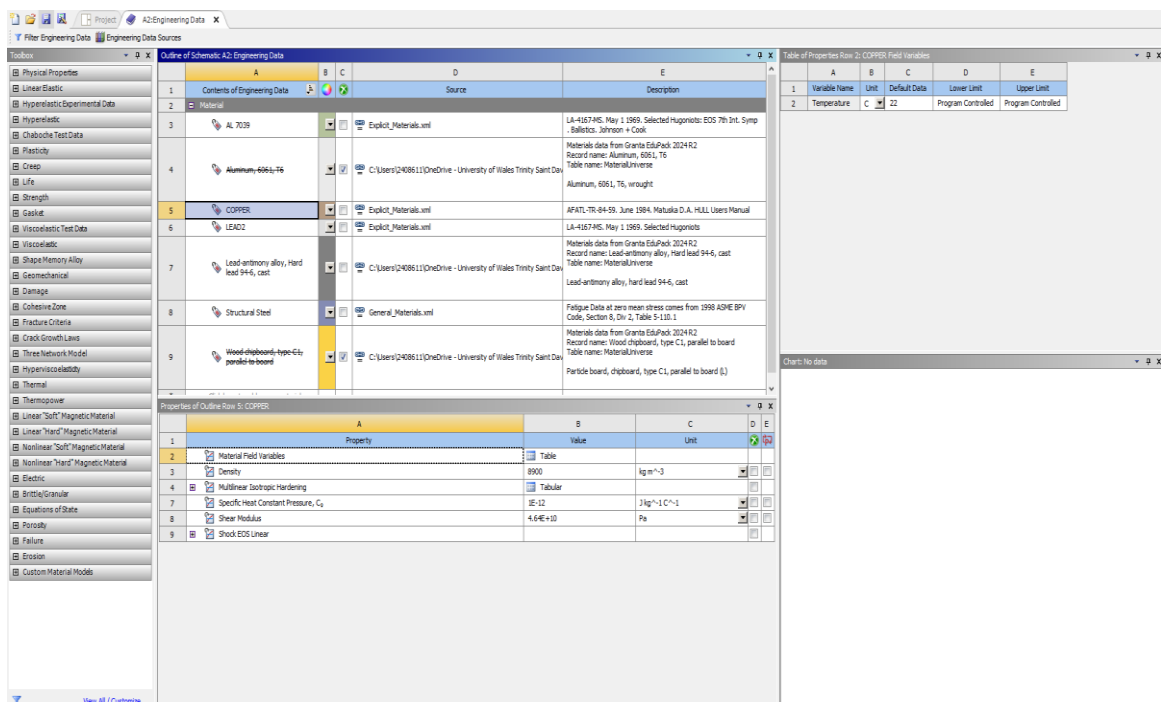


Figure 4 - Imported Materials list

From this list it can be seen that other forms of aluminium as well as wood chipboard were explored as potential variables for the plate, however it was later discovered that these did not contain the relevant information therefore were not used and the material properties were suppressed from the project to avoid confusion, another helpful tool from the software.

Once all the materials had been assigned to the project, the engineering data function was closed and the model function was opened, which opened the solver in a separate window. The solver allows the model, setup, solution and results to be generated and saved to the project at once, without having to return to workbench. Using the solver, the materials were assigned to the parts in the model, with the variation in plate material as shown below:

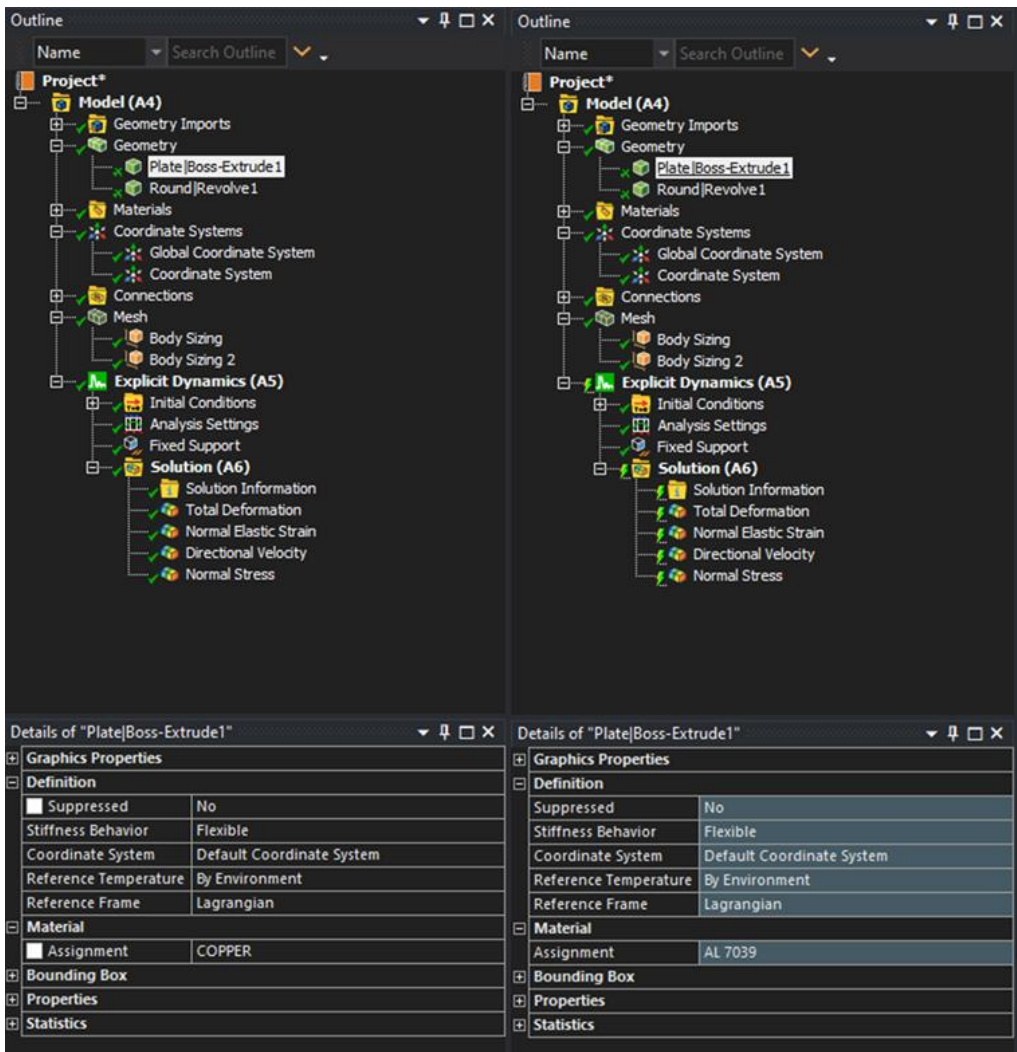


Figure 5 - Material Selection for Plate

2.1.2 Coordinate Systems

The Coordinate system within the solver is a way of determining load points in a project, based on coordinate origin.

It was quickly discovered that the imported model was created slightly off centre from the global coordinate system. As a result, a second coordinate system was created to make any loads central to the model, this can be seen in figure 6.

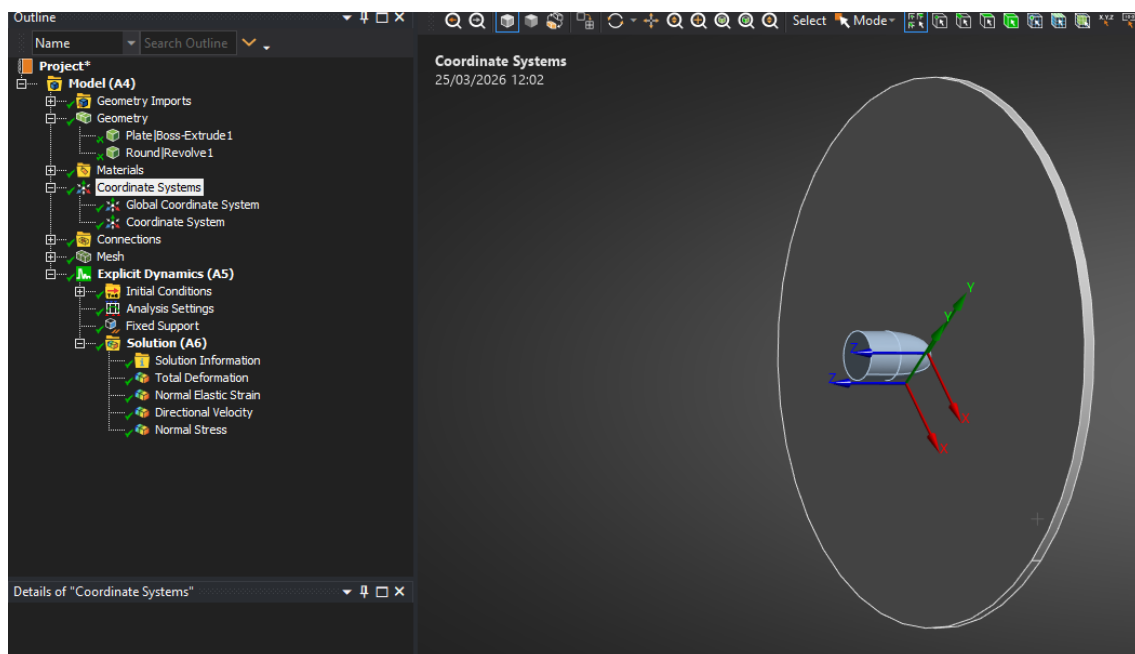


Figure 6 - Coordinate System Creation

2.1.3 Mesh Generation

Meshing of the model is important for a number of reasons, directly affecting the level of accuracy as well as the computing power required from the simulation, where a finer mesh creates more elements.

A global mesh of 3.5mm per element size was set for this simulation.

Whilst global meshing is the easiest way to mesh a model, a useful tool to reduce computing requirements is the sphere of influence tool. What this allows is an area where the mesh can be independently controlled, to allow fine analysis of specific areas of interest or where there may be weak points. For this simulation, a sphere of influence was created for area around the site of impact between the

bullet and the plate, centred on the coordinate system created in the centre of the model, and only assigned to the plate as seen in figure 7.

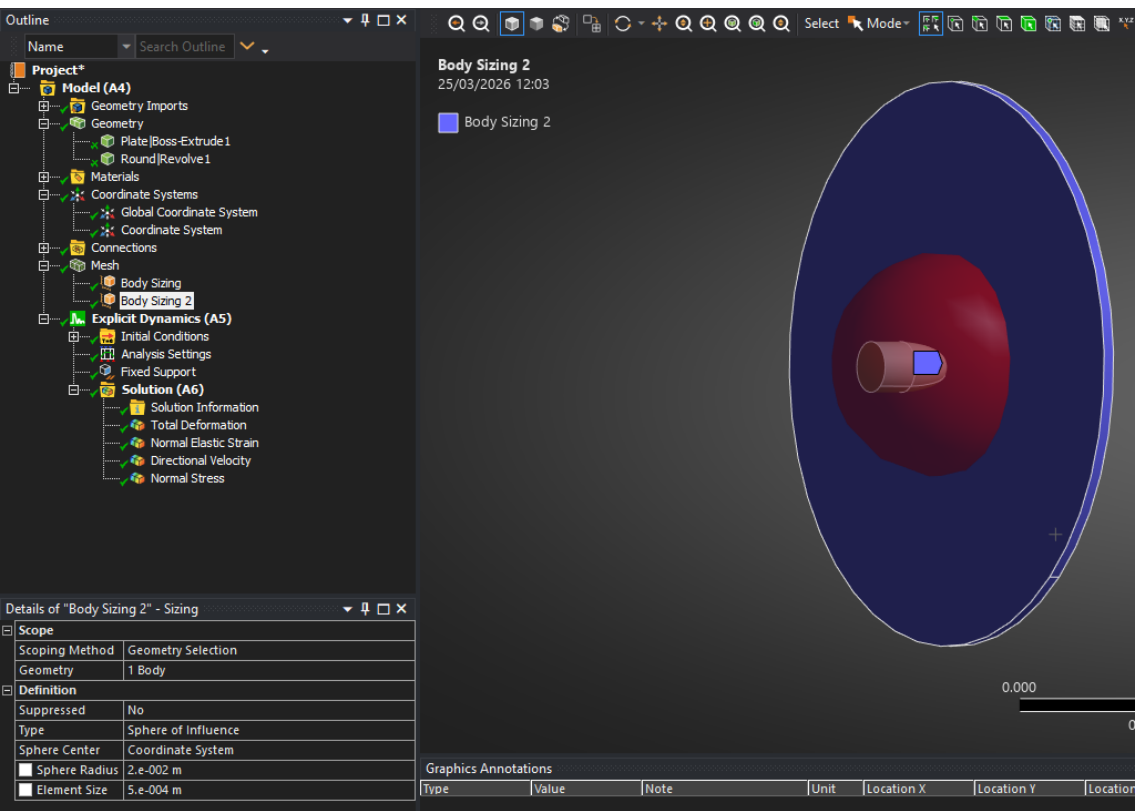


Figure 7 - Sphere of Influence

The radius for this can be seen at 2cm, with the element size within this sphere reduced to 0.5mm. This allowed greater accuracy of element distortion in the plate for the area affected by the impact, as well as reducing computing power requirements, as the resultant number of element nodes were 5,104. If the entire plate had been globally meshed at 0.5mm then the element node count would have increased to 31,416, over six times that of the nodes generated, requiring significantly more computing power to solve.

The resultant mesh can be seen in figure 8.

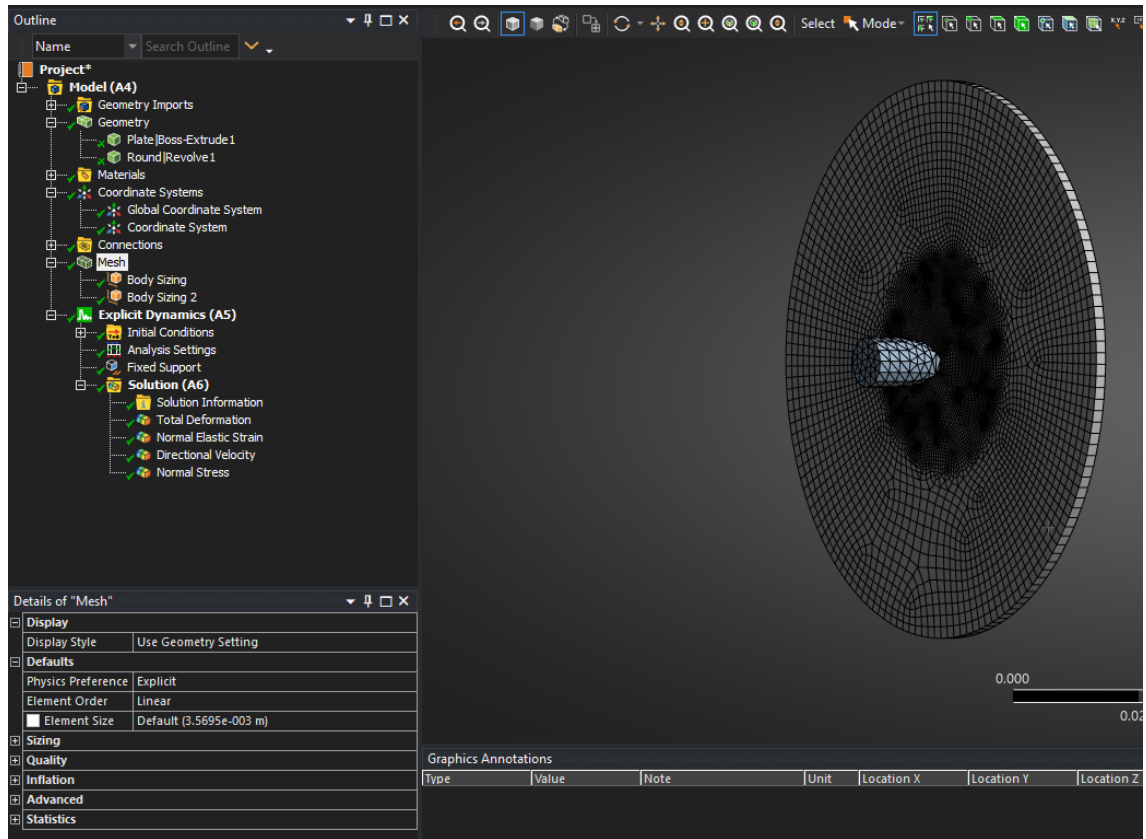


Figure 8 - Meshed Model Including Sphere of Influence

2.1.4 Initial Conditions

Once the materials were set and the model meshed, the initial conditions were then set for the simulation. This included a load of inertia of the bullet only of 300m/s on a vector directly towards the plate. Other initial conditions included ambient room temperature for the environment as well as fixing the plate by the front edge of the outer face, preventing any inertial transfer of velocity of the plate so that the full force of the impact would be evident in the results. This can be seen in figure 9.

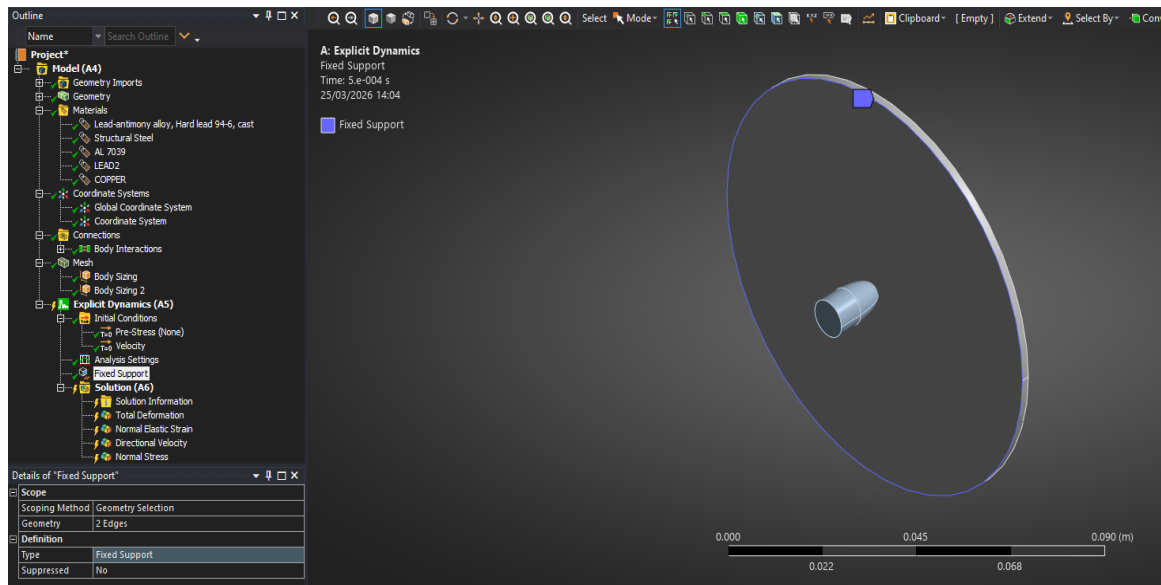


Figure 9 - Edge Fixture as Part of Initial Conditions

2.1.5 Analysis Settings

For the analysis, four different aspects were identified and set prior to solving:

- Deformation
- Elastic Strain
- Stress
- Velocity

Due to the program being set up as an explicit dynamic solver, this provided the most information possible for analysis.

If more analysis information was required, various subroutines and feedback loops could have been incorporated into the workbench area of Ansys, such as thermal loads and state of change due to any reactions during material impact, however this would have made the simulation run for quite some time as it would require much more computing power.

During the execution of the solver, an estimated time until completion can be seen on the solution information tab. This tab also shows the real time passed in the simulation for each solve step, and the time increments between calculations, as per figure 10.

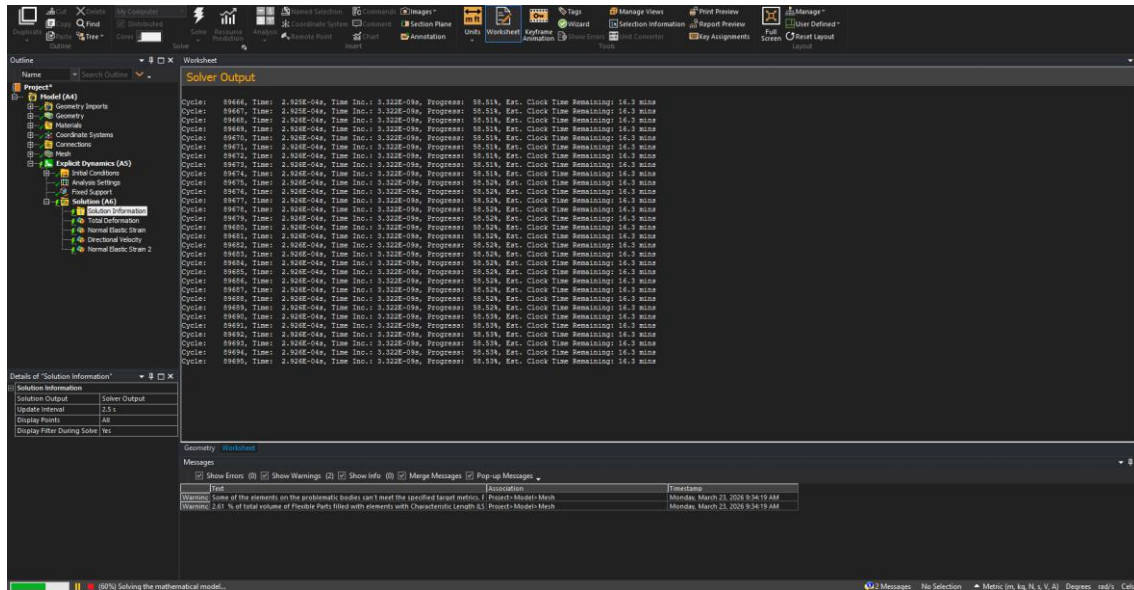


Figure 10 - Solution Progress Monitor

This was also important to track as the timestep could be analysed against the level of mesh refinement, ensuring the number of calculations and therefore time until completion was realistic, as the time steps are directly proportional to the smallest mesh element, therefore a smaller mesh element would result in a decrease in timesteps and increase in solve time.

2.1.6 Results

As previously mentioned, the solver generated results into four main categories:

- Deformation
- Elastic Strain
- Stress
- Velocity

This allows analysis of all major mechanical components of the materials used for the plate, when impacted by a standard 9mm projectile at 300m/s.

2.1.6.1 Deformation

Deformation of both the aluminium and copper can be compared to understand how the materials respond to the impact. Figures 11-16 show the point of projectile impact, first penetration and final state of the material plate.

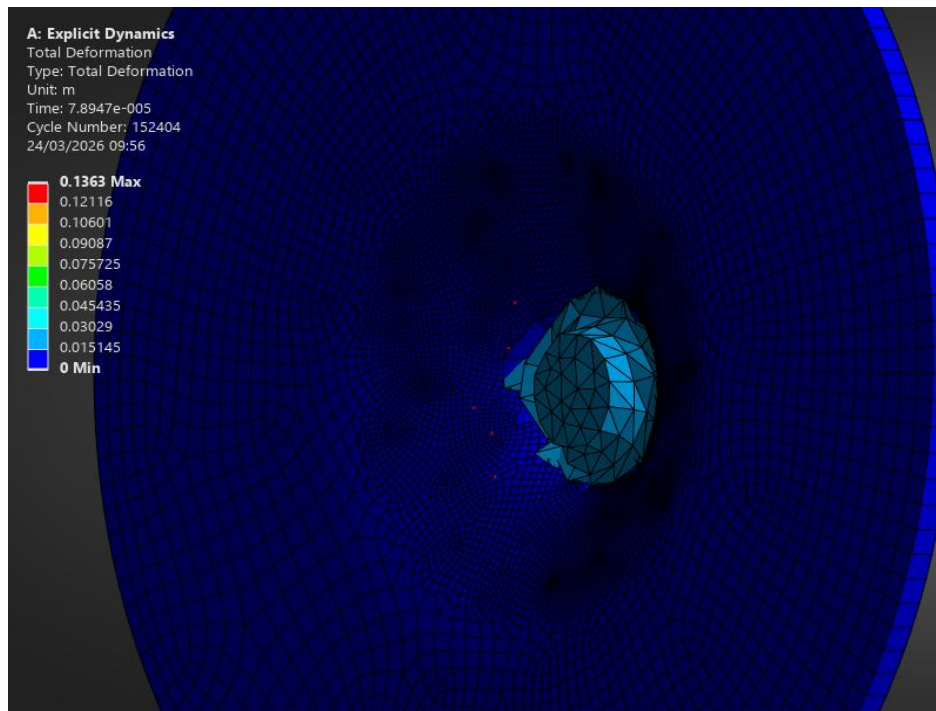


Figure 11 - Initial Impact of Projectile on Aluminium Plate

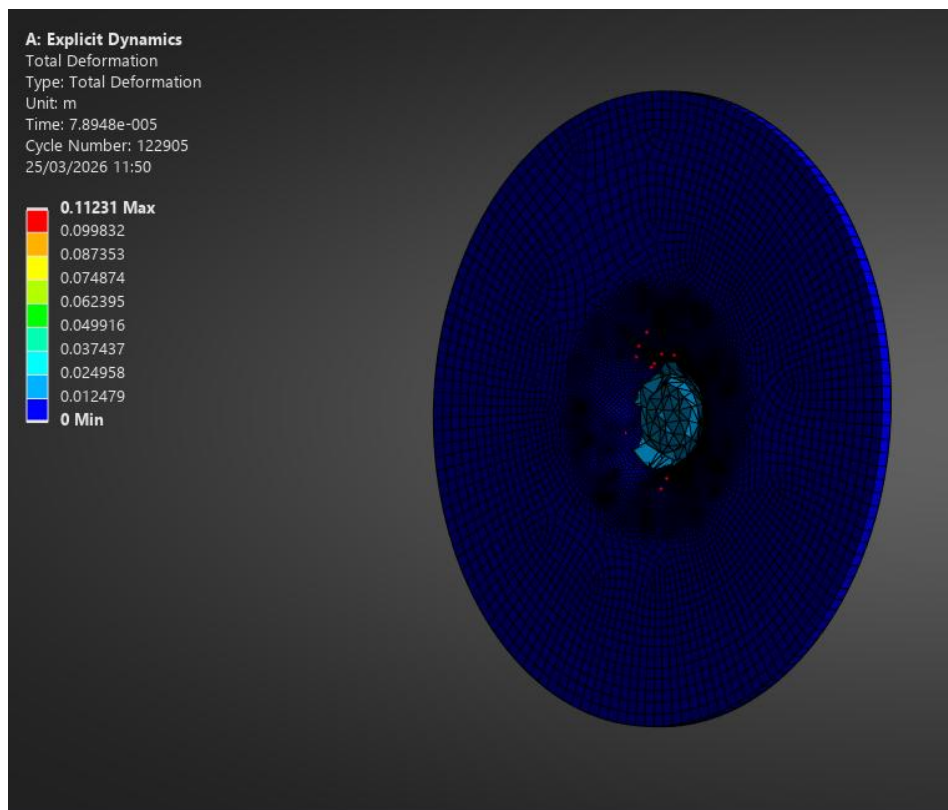


Figure 12 - Initial Impact of Projectile on Copper Plate

What can be surmised upon initial impact is that whilst the copper deforms more locally to the projectile, the aluminium exhibits a wider area of deformation within the plate. This is expected due to the copper being a more ductile and malleable material compared to the aluminium, which is more brittle. As a result of this, the red dots in the images represent element nodes which have been deformed by too much, and therefore deleted by the program, it can be seen that many more have been deleted for the copper.

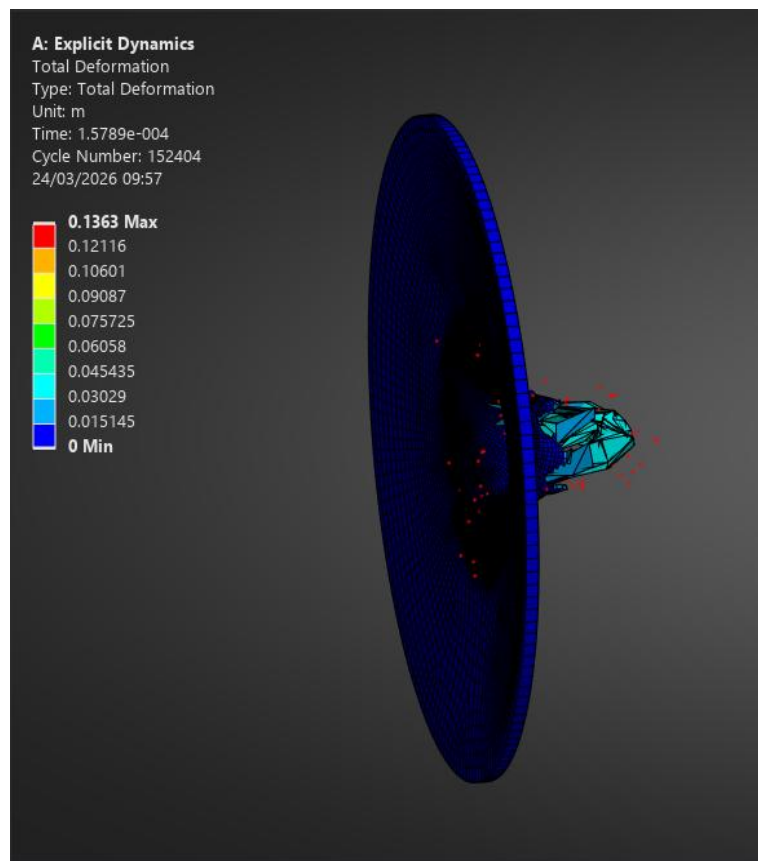


Figure 13 - First Penetration of Aluminium Plate

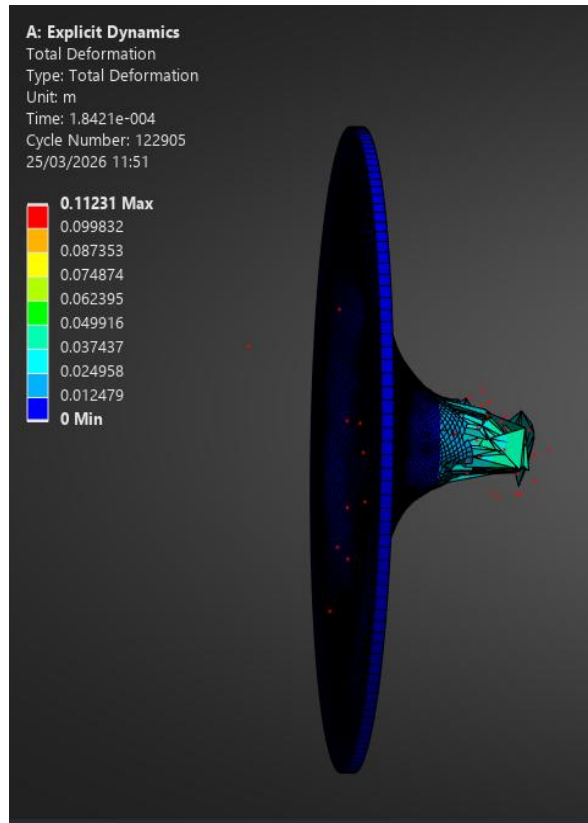


Figure 14 - First Penetration of Copper Plate

Upon first penetration, the previously seen deep, localised deformation of the copper plate has deepened further, whilst the area of deformation in the aluminium has also continued to spread, giving the aluminium a deformation spread almost all the way to the edge.

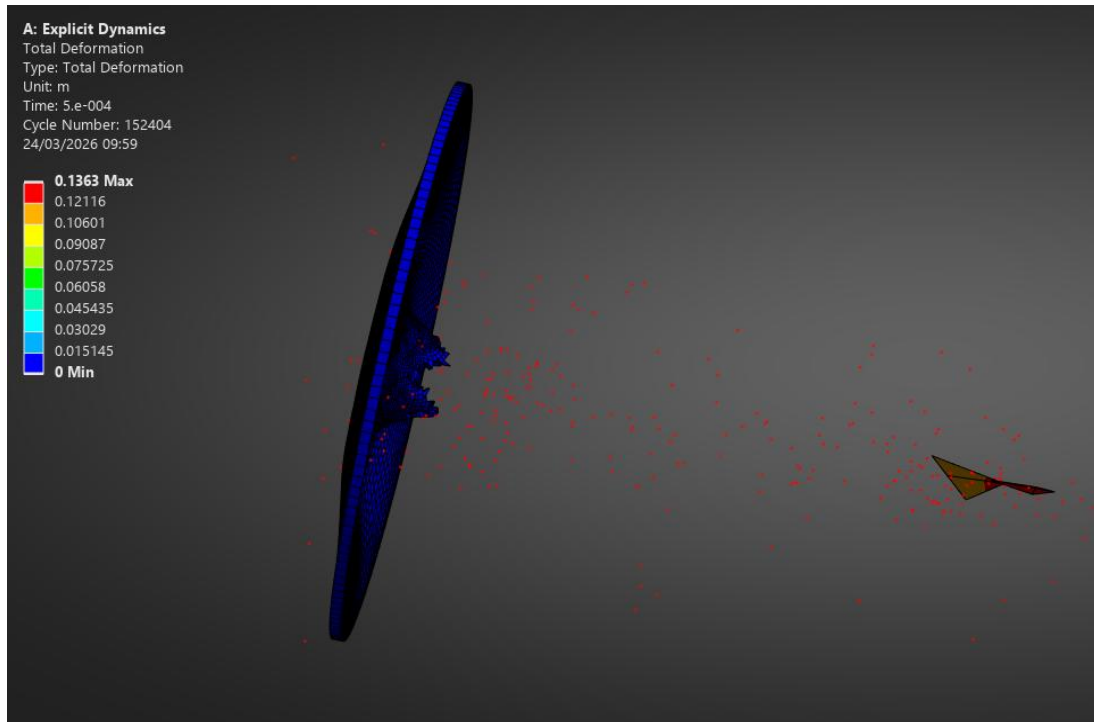


Figure 15 - Final State of Aluminium Plate

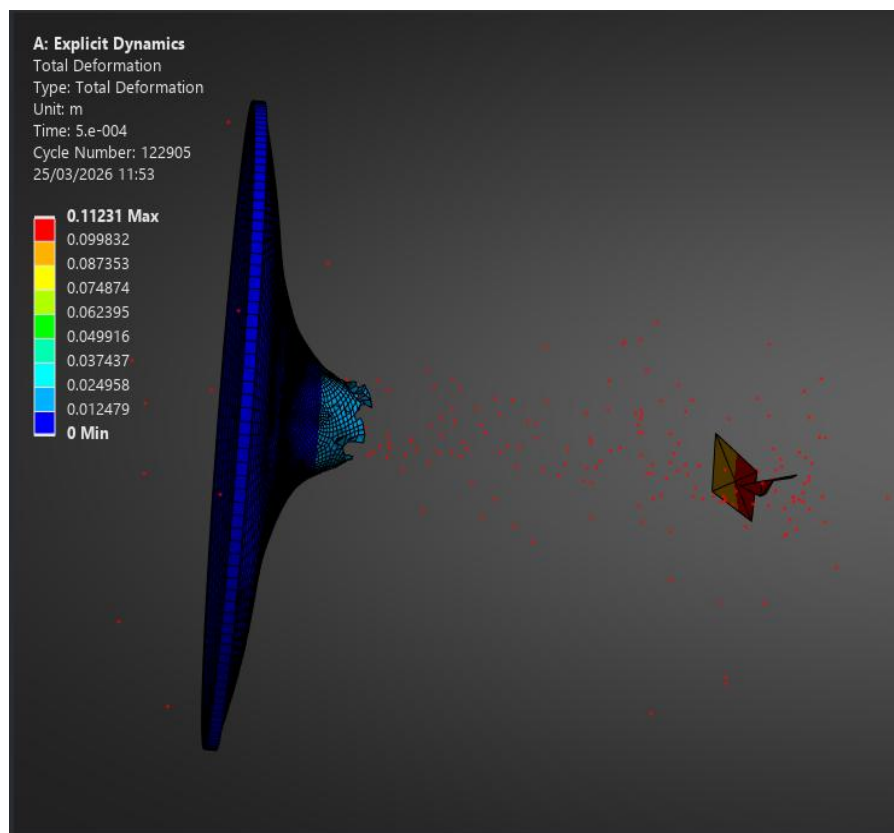


Figure 16 - Final State of Copper Plate

The final state of materials showed a continuation of the trends discussed before, with a reflection wave being exhibited by the aluminium providing a more even deformation in both the forward and negative plane of travel in reference to the vector of the projectile. The copper plate resulted in a deeper, localised cone shaped result. Something else also exhibited was the destruction and deletion of the majority of the element nodes of the projectile due to excessive distortion, which, being made of lead antimony, represents what the projectile is designed to do, which is to break apart and fragment upon impact with a target to ensure the maximum transfer of kinetic energy into the impacted target. Although the complete deletion of nodes is also a limitation, as the kinematics of those element nodes after they break down cannot be accurately analysed to understand any further kinetic energy transfer rates or velocities of fragmentations.

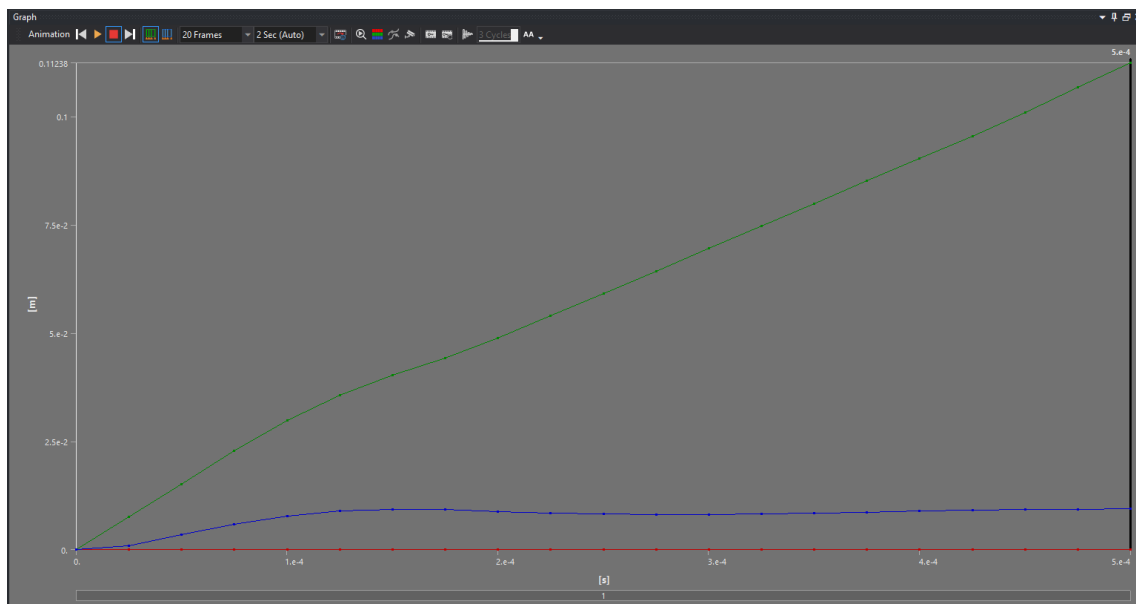


Figure 17 - Deformation Graph of Aluminium

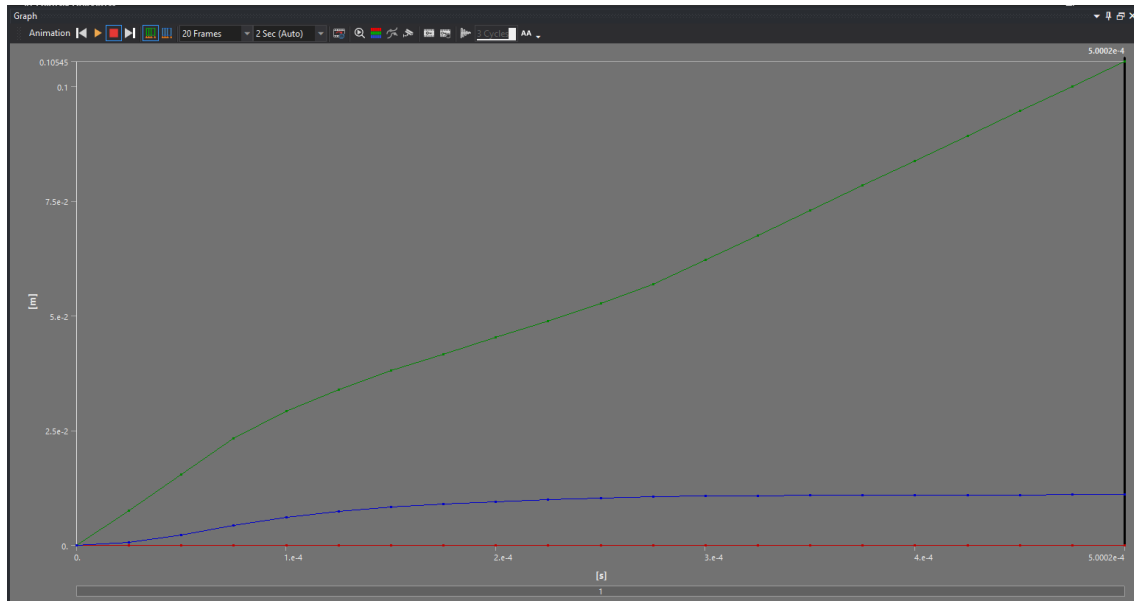


Figure 18 - Deformation Graph of Copper

The deformation graphs show that whilst certain elements of the copper may have deformed more than in the aluminium, the overall deformation of the whole material was greater in the aluminium, with red being the minimum, blue being the average and green being the maximum deformation of elements in the simulation.

2.1.6.2 Elastic Strain

For elastic strain, the focus was on analysing when the elastic strain was the greatest within the materials, as seen in figures 19 and 20.

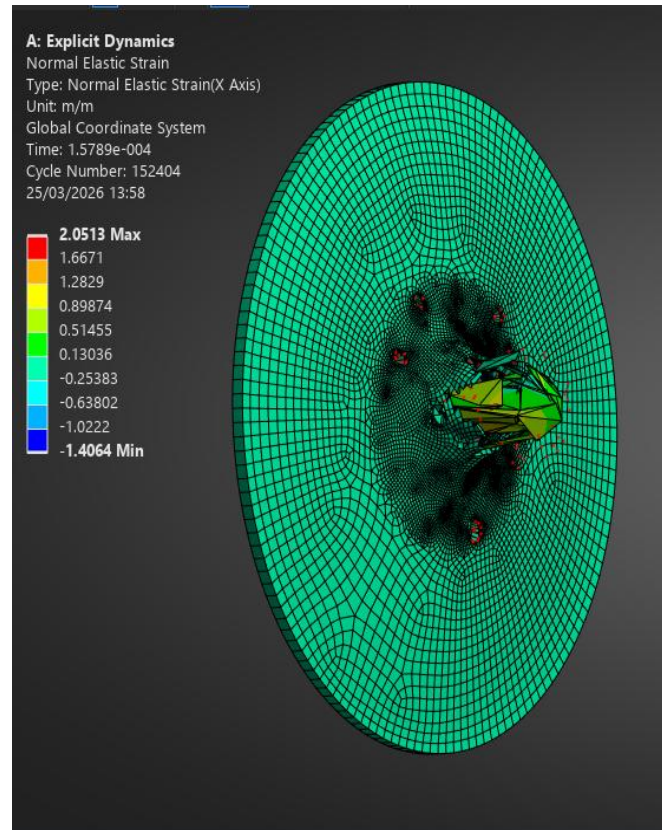


Figure 19 - Elastic Strain in Aluminium

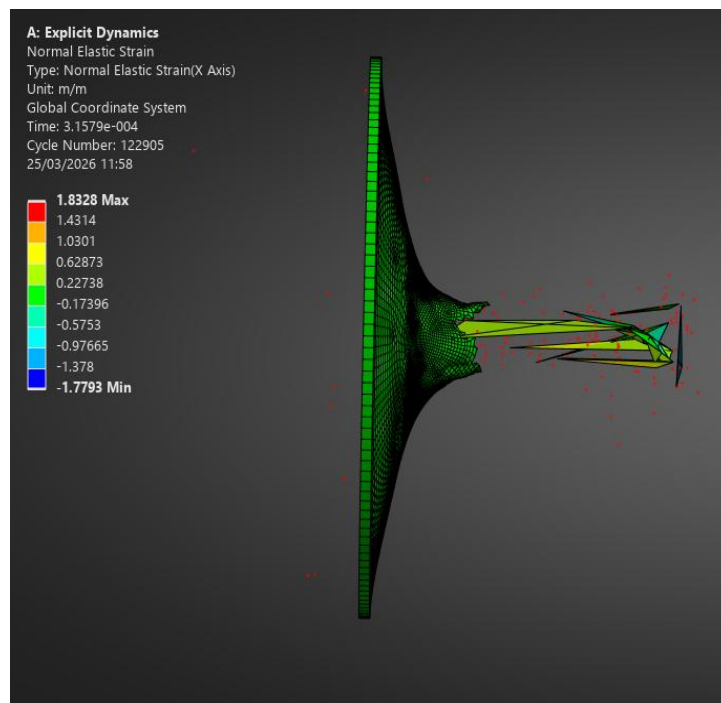


Figure 20 - Elastic Strain in Copper

These figures show the maximum strain not only presented at different points during the simulation, but also the values of maximum strain were different for the different material, with aluminium having a higher peak value much sooner in the simulation. This is validated by the information presented in the graphs in figures 21 and 22.

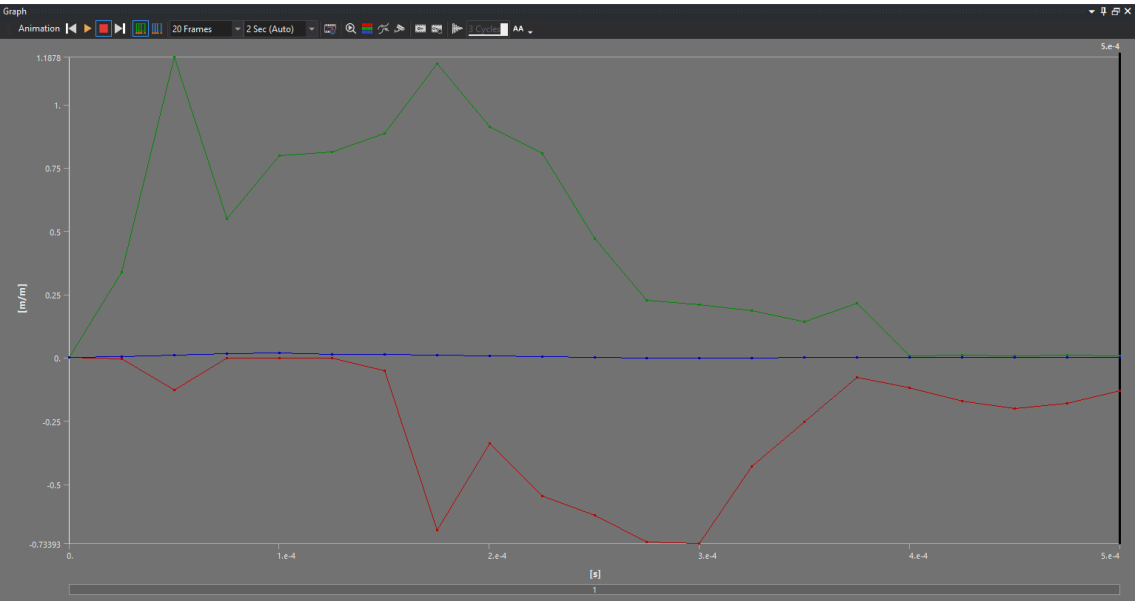


Figure 21 - Elastic Strain Graph for Aluminium

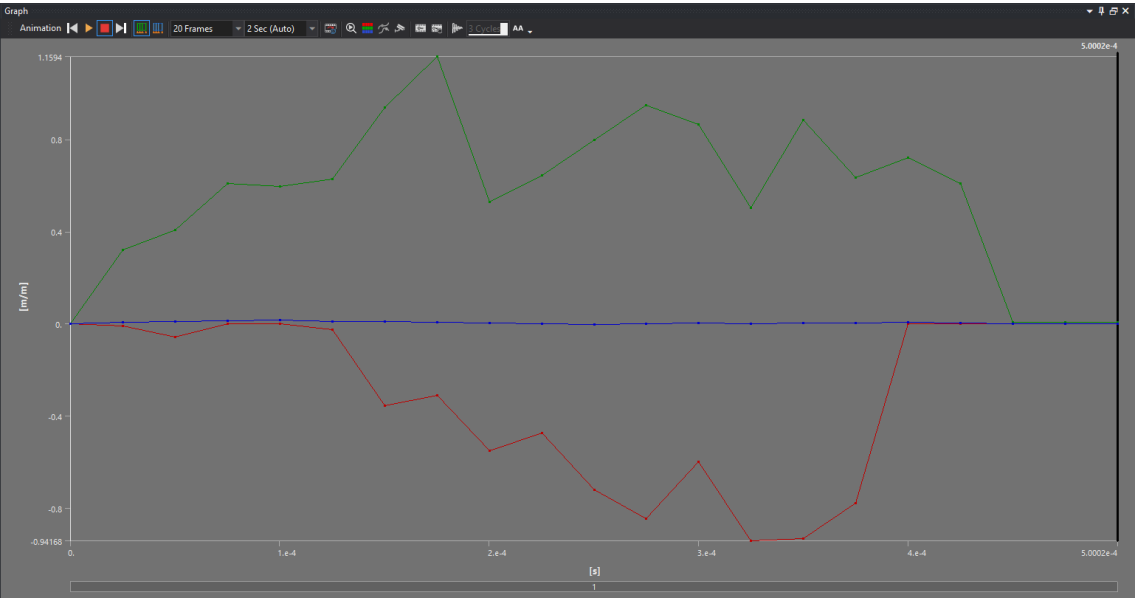


Figure 22 - Elastic Strain Graph for Copper

2.1.6.3 Stress

Similarly to how the elastic strain had been analysed, the analysis of the stress in the material can be seen in figures 23 and 24, at peak stress during the simulation.

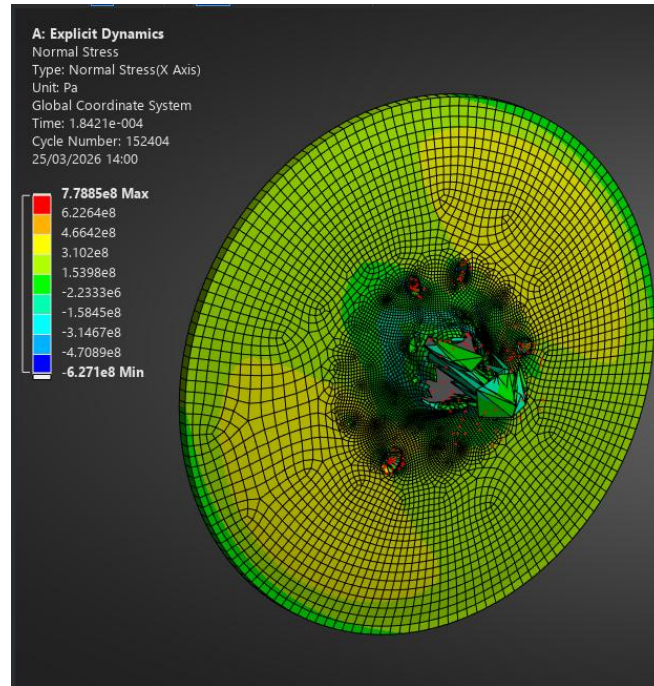


Figure 23 - Stress in Aluminium

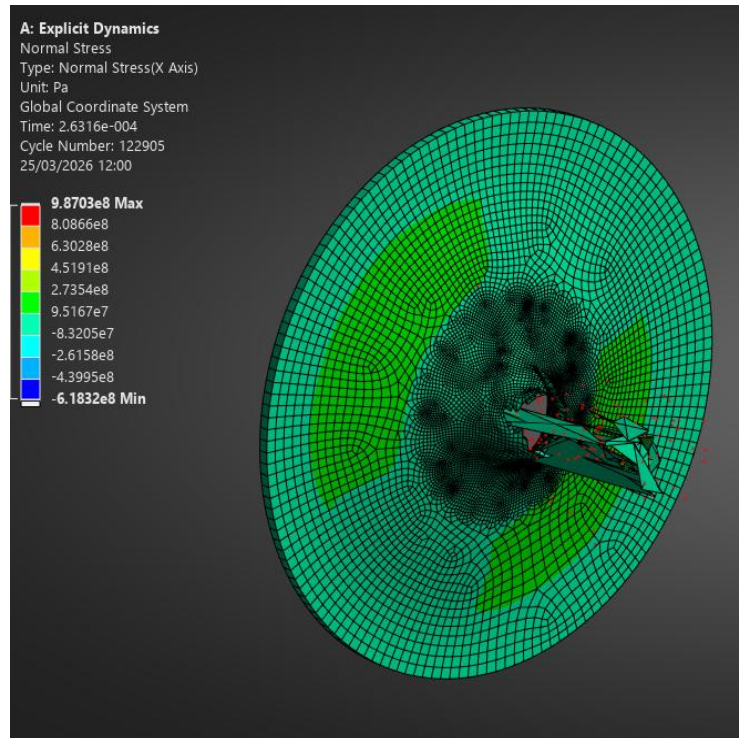


Figure 24 - Stress in Copper

What is more interesting is that whilst the stress peaks significantly higher through the aluminium, the stress dissipates more uniformly from the copper, as a result of its ductility, as represented in the graphs seen in figures 25 and 26.

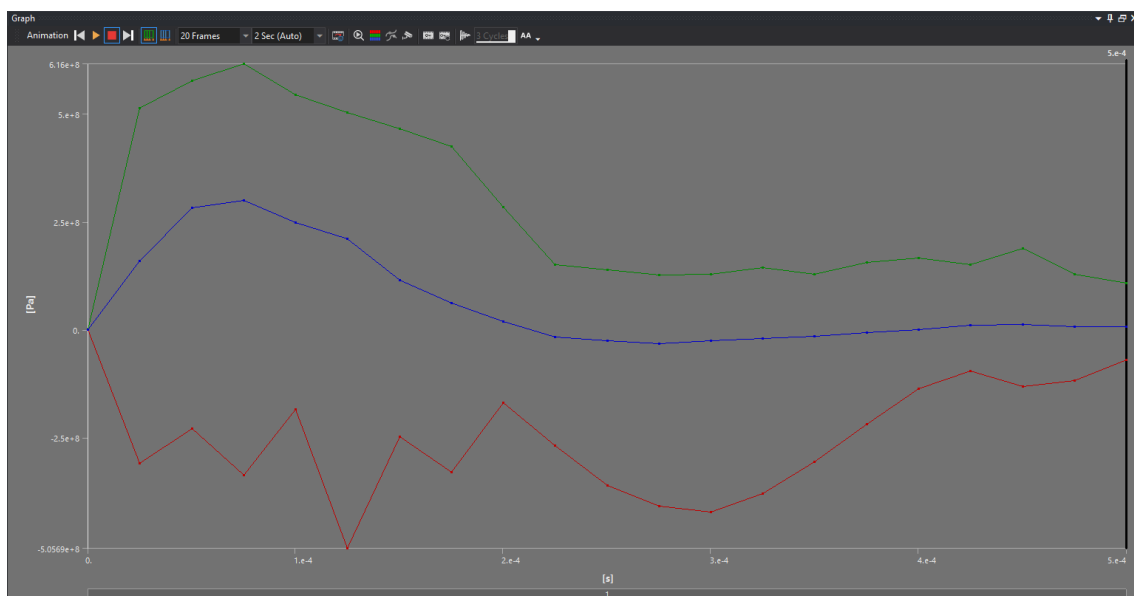


Figure 25 - Stress Graph for Aluminium

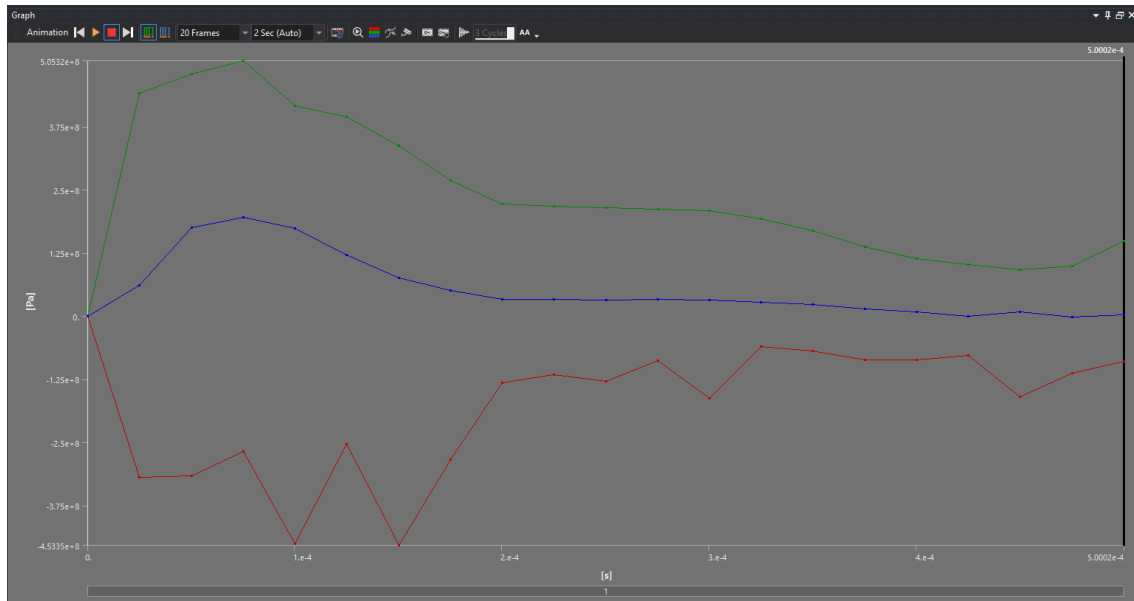


Figure 26 - Stress Graph for Copper

2.1.6.4 Velocity

For velocity, it was much more difficult to discern by analysing the visual simulation due to its high speed and small changes, therefore a better representation has been presented in the form of graphs in figures 27 and 28.

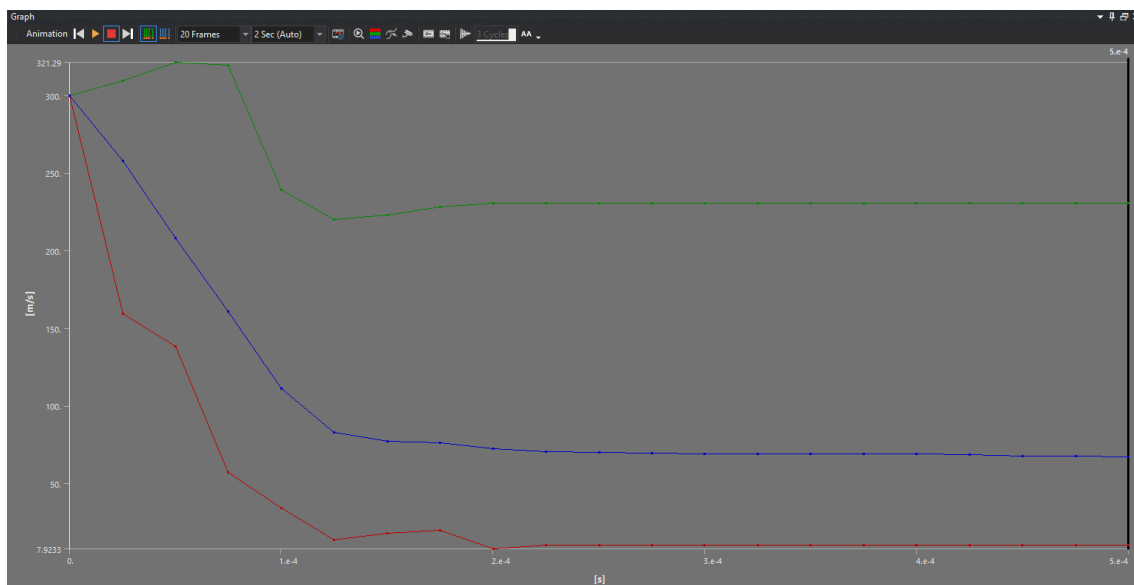


Figure 27 - Velocity Graph of Projectile for Impact on Aluminium plate

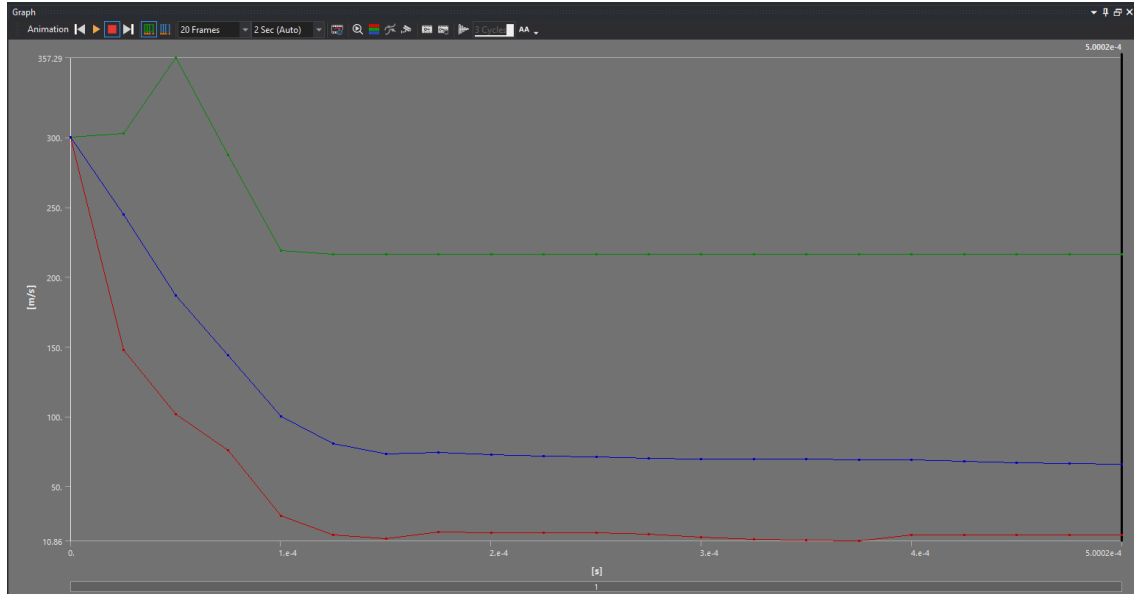


Figure 28 - Velocity Graph of Projectile for Impact on Copper Plate

These graphs provide representation of the velocity of the projectile throughout the simulation, where the green line in the graph represents maximum velocity shows that copper, being a more malleable material, provides less resistance initially but the prolonged interaction between the projectile and the plate leads to a greater deceleration before the projectile has punched through in its entirety.

2.1.7 Mesh Independence Study

Mesh independence is described as the point in which refining the mesh further does not heed results which are considered viably efficient, when taking into account resource and time requirements to run such simulations [5].

For this simulation, where aluminium was used for the plate material, mesh independence was initially measured as the point in which the projectile penetrated the plate, known as mesh convergence, and further refined to understand which point would not generate significantly better results. This can be seen across figures 29 to 32, where the sphere of influence was reduced, but also the mesh element size for both the sphere of influence and the entire body were increased initially, then refined until initial penetration and then until a clean penetration was achieved.

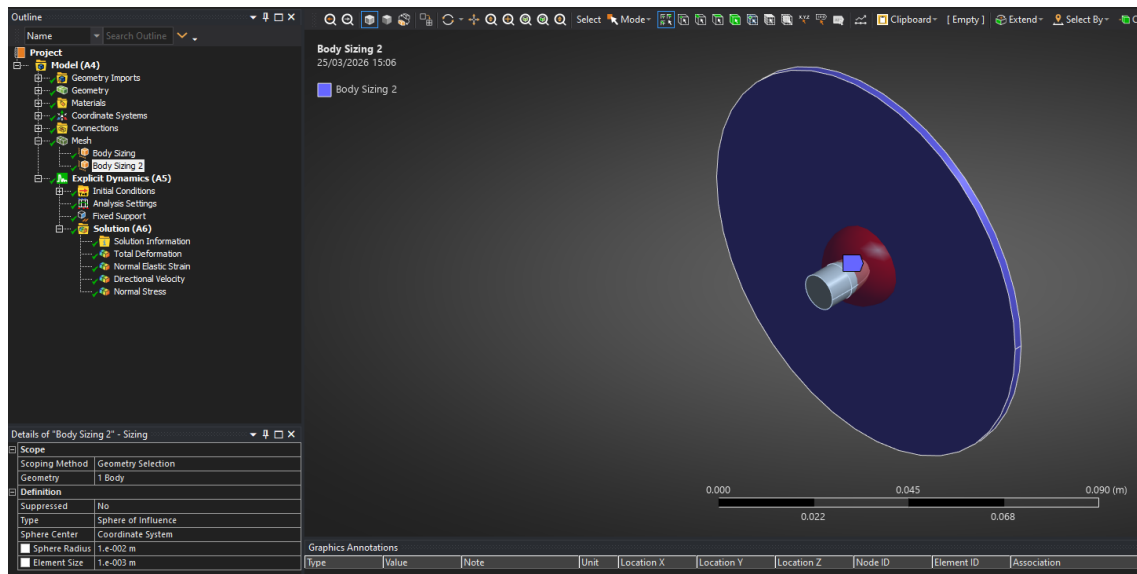


Figure 29 - Sphere of Influence Reduction and Mesh Element Size Increase

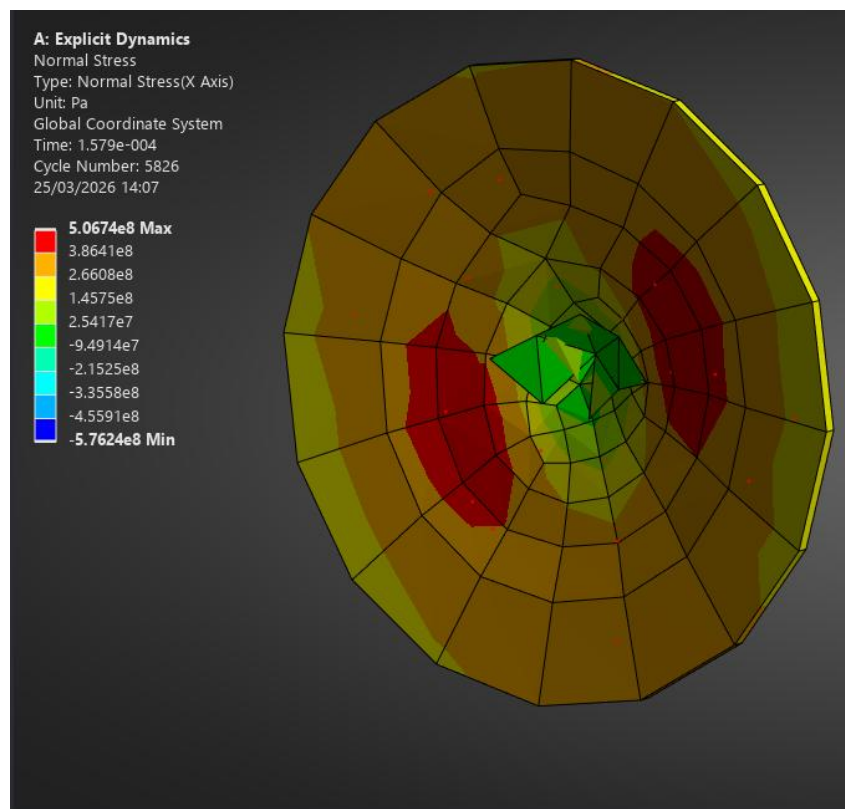


Figure 30 - Non-Penetration of Mesh Due To Element Size

For figure 29 and 30, above, the mesh element size for the body plate was increased to 35mm and the sphere of influence, whilst itself decreased to 10mm,

the element size was increased to 1mm, as a result the mesh elements were too large and therefore the projectile didn't penetrate the plate, however, large areas of high stress can also be observed. It can also be noted that the entire simulation solved in around 4 seconds flat.

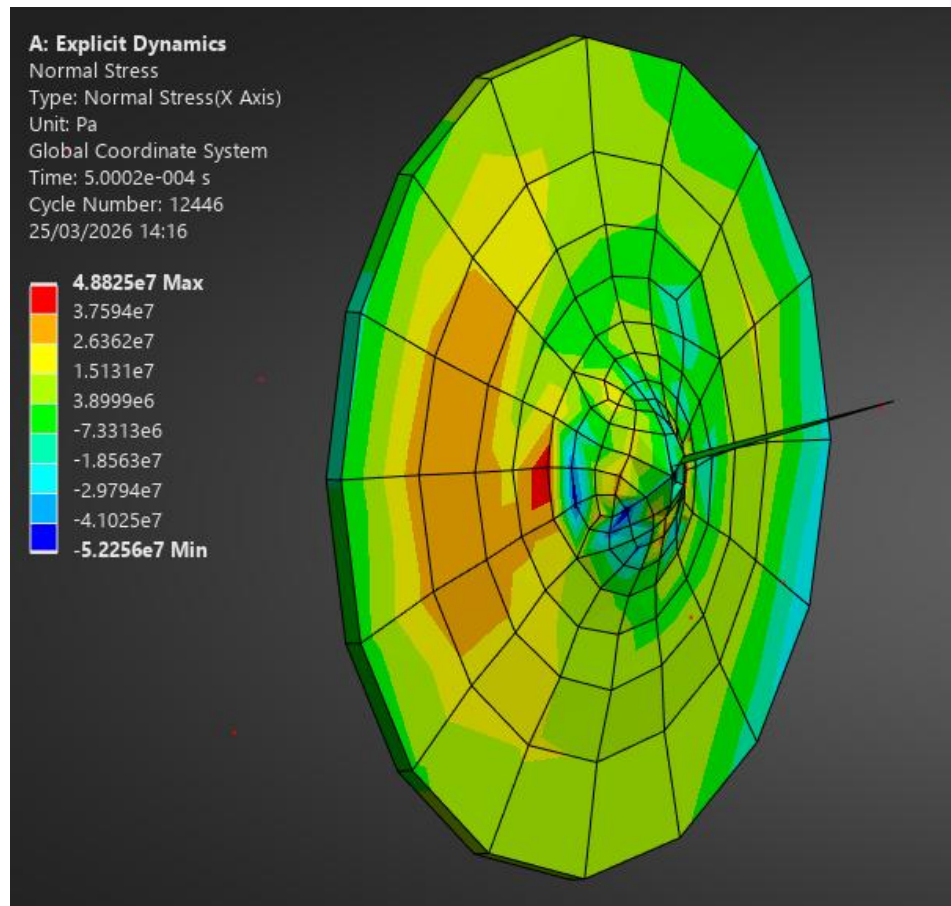


Figure 31 - Mesh Convergence Point

For figure 31, the point of mesh convergence can be seen where the mesh element size for the body was decreased to 20mm, with the sphere of influence staying the same diameter but the mesh element size reducing to 2mm. Slight penetration can be seen here, with the areas of stress more intricately refined, starting to provide the results in which the simulation was aiming to achieve.

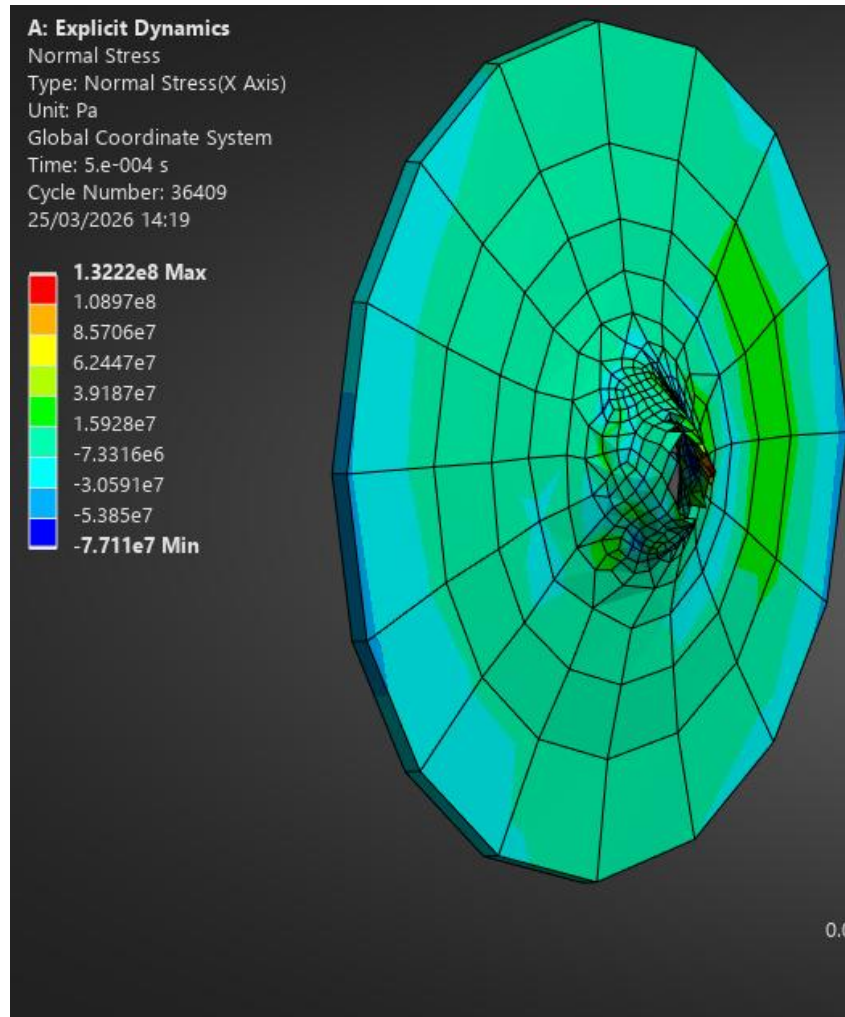


Figure 32 - Mesh Independence Point

In figure 32, the mesh independence point had been achieved by further reducing the sphere of influence mesh element size to 1mm. This provided a clean penetration and a more refined stress map across the plate. As the element size had decreased, the level of detail allowed the simulation to understand better how the stress dissipated across the plate, showing overall less stress throughout the material. A further refinement of 0.5mm mesh element size increased accuracy by <5% and therefore can be considered negligible as it increased the solve time by a further 42 minutes, making the trade-off inefficient.

This result was solved in about 3 minutes, whereas the simulations seen earlier in the report in the 'Results' section took around 45 minutes to solve. This

highlights the importance of mesh independence in order to achieve a higher efficiency, through quicker solves, by not requiring as much computing power.

2.1.8 Boundary Condition Influence

Boundary conditions are constraints which define a number of factors within CAE, and are formed of constraints, loads and environmental conditions and interactions which may affect how a structure or fluid behaves [6].

One boundary condition change for the simulation was the fixture point of the plate, where the plate had previously been fixed by the front edge throughout previous simulations.

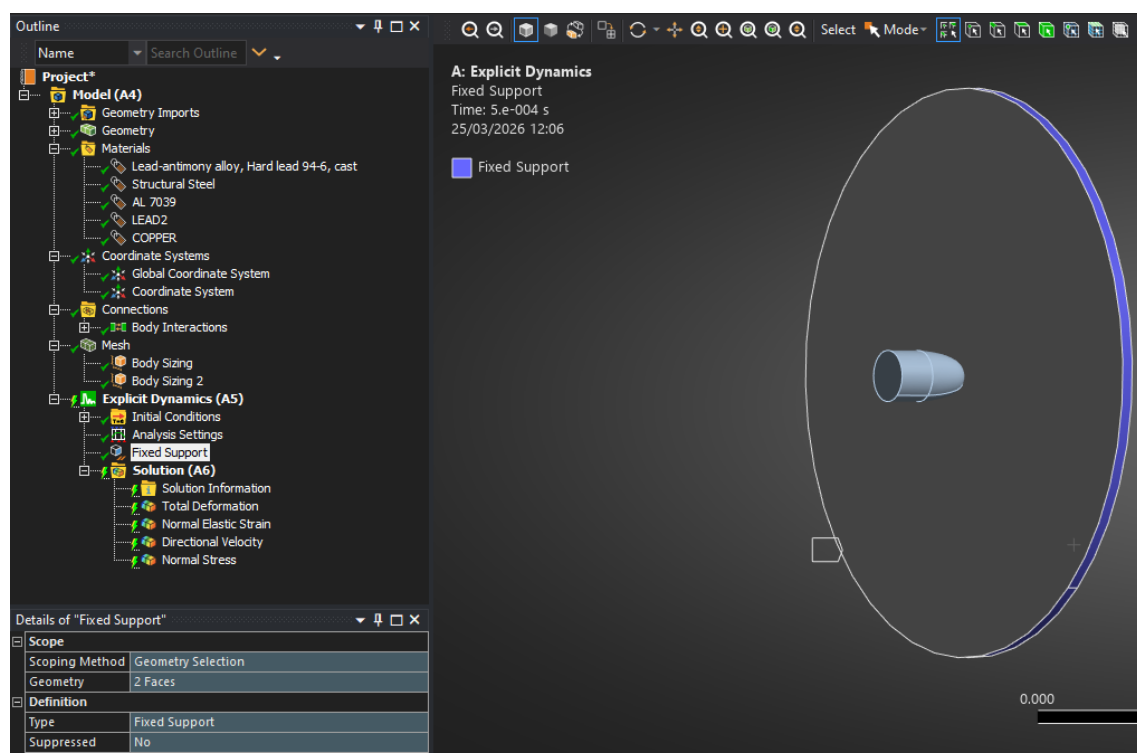


Figure 33 - Change of Fixture of Plate

In Figure 33, the aforementioned boundary condition was adjusted so that the entire outer face was fixed, rather than just the front edge.

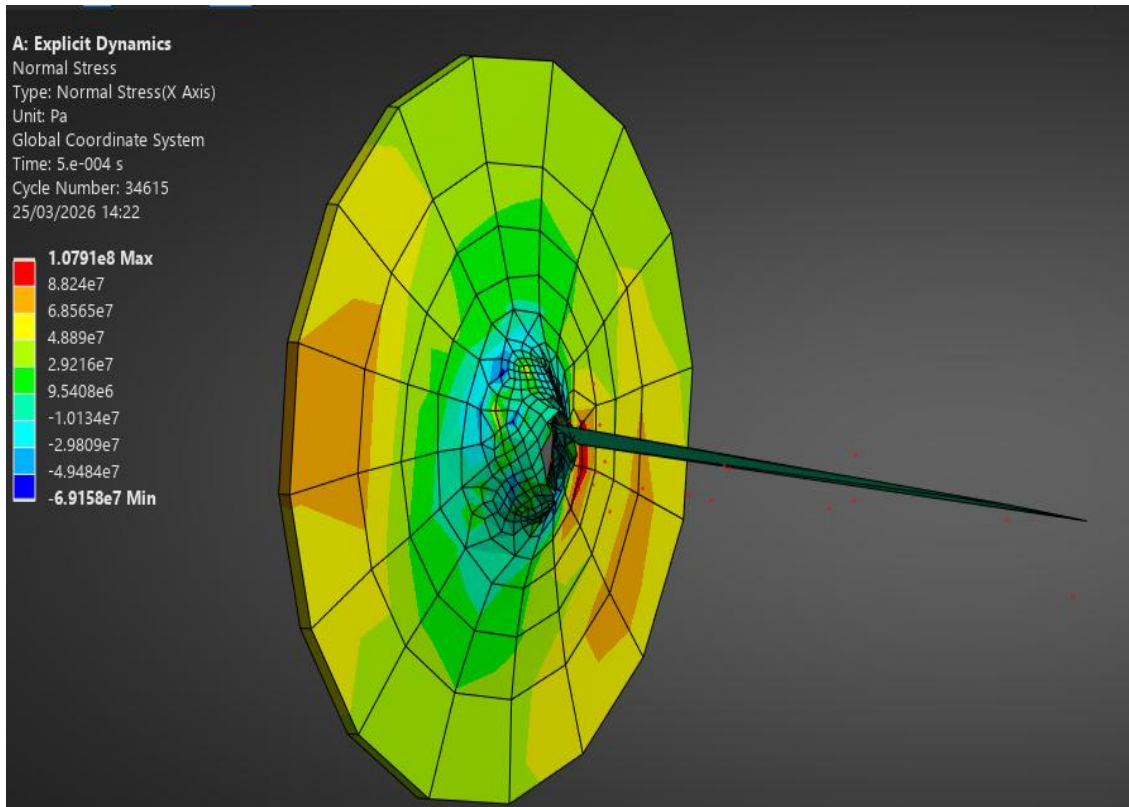


Figure 34 - Fixed Face Result

The result of fixing the entire face can be seen above in figure 34, where higher stresses can be seen closer to the edge of the plate, as well as a distorted node as a result of a slight change in deflection, due to the plate not having as much freedom to flex under pressure.

2.2 Validation

Validation, sometimes referred to as verification and validation, is the study in which a set of simulated computer results are analysed and compared against similar computer models as well as real life test results to understand the accuracy of the simulation and ensure the results are as close to what can be expected as possible, providing reputable resource at a greater efficiency. It is defined by ASME V&V 10.1 Standard as: “*substantiation that a computerized model within its domain of applicability possesses a satisfactory range of accuracy consistent with the intended application*” [7].

2.2.1 Validation of Aluminium Plate

For validation of the aluminium plate simulation, a comparison against similar scenarios shows the same deformation pattern, as can be seen in figure 35.

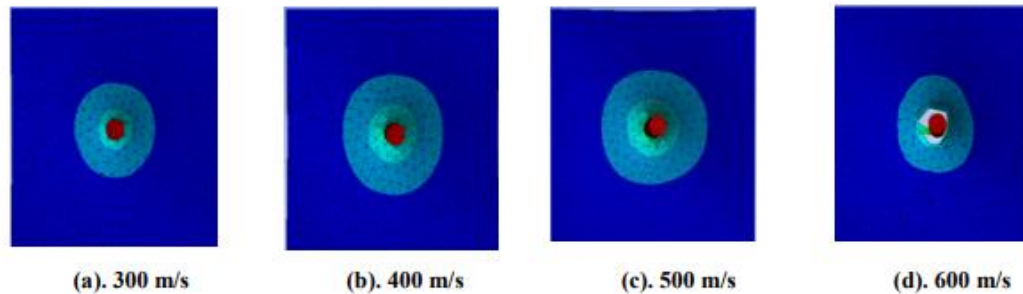


Figure 35 - "Bullet impact and penetration of target plate at normal direction at various velocities". Adapted from [8].

What can be seen in this image is the various outcomes of a similar simulation, with the only defining difference being the material used is AL2024 T3 Aluminium, of which properties can be seen in Table 1.

Table 1 – Comparison Table of Properties of Aluminium

Property	AL2024 T3 [8]	7039 [9,10]
Density (Kg/m ³)	2780	2740
Ultimate Tensile Strength (MPa)	483	366
Tensile Yield Strength (MPa)	345	286

The values of each material variation show that the AL2024 T3 Aluminium has slightly higher density yet considerably higher tensile strength, which explains why, at the same velocity of impact, there is not a complete penetration in Figure 35, and in fact it takes twice the velocity to completely penetrate [8].

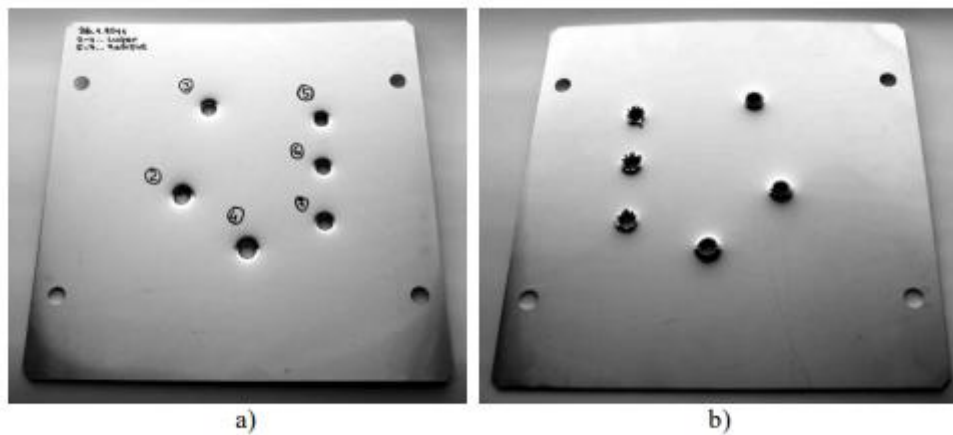


Figure 36 - "Sheet metal sample penetrated by a 9mm Luger bullet with full metal jacket of standard velocity, see shot No. 3 and 4 from impact side (a) and output side (b)". Adapted from [11].

Figure 36 provides real-world validation of the final deformation of material, where the same size projectile was used to fire at 2024T3 Aluminium, however this material is only 1.2mm thick, the projectile consisted of a full metal jacket of copper with a lead core, and the velocity in which the projectile was fired was slightly higher at 396 m/s for (a) and 387 m/s for (b) [11], compared to the case study of a 2mm thick plate, a complete lead antimony projectile and an initial velocity of 300m/s used in the case study.

Although these parameters are different to the simulation parameters in the case study, the same petalling has been exhibited in both, as expected in aluminium, and as such provides validation that the results shown in the case study compared to what could be expected in the real world should be considered acceptable. If the real-world test case conditions were to match that of the case study, then it could be expected that the results would be accurately similar for both.

2.2.2 Validation of Copper Plate

The validation of using copper plate was much more difficult to ascertain, as not many examples exist simply because copper is not genuinely used in many real world applications due to its heavy weight and softness compared to alternatives such as steel or ceramic, but it is used as a jacket in bullet components to assist

the penetration of solid targets such as car doors, although this was only briefly mentioned in sources and no specific examples were given.

With that said, one such example does exist, where a steel bearing 12mm in diameter had been studied to understand the difference in effects on copper and steel plating. Figure 37 shows the results of the copper plate, which was also 2mm thick.

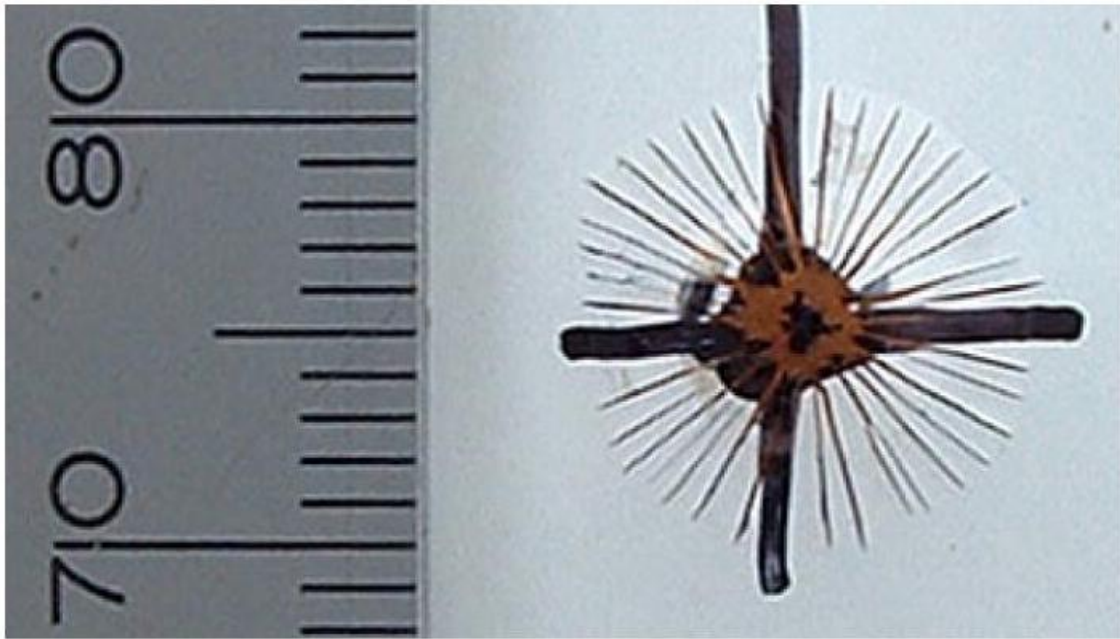


Figure 37 - "Typical paint detachment patterns (post-impact): (a) radial cracks propagated to a maximum radius for 2 mm thick copper plates". Adapted from [12].

This real-world example shows similar plugging (or ductile enlargement) as to what has been seen in the case study simulation, however the severity of the plugging and clearance is slightly different as a result of the projectile being of a different shape, size and mass, as well as the projectile being fired at a significantly slower velocity of 70m/s. The rear surface of the plate was also sprayed with matt white in the real-world example, to more accurately highlight deformation, which can be seen in the form of radial cracks in the image in figure 37 [12].

This provides validation of the results in the case study simulation as they are accurately similar, with only minor differences due to variations in the model and initial conditions.

3 Analytical and Software Comparison

Although Ansys Workbench has been used for the purpose of the case study, it is important to understand other dynamic solver software available, including advantages and disadvantages as well as applications of each type of solver. A comparison between Ansys Workbench, Dassault Systèmes Abaqus and Autodesk fusion 360 can be seen, as two of the industry leaders in dynamic solvers.

3.1 Ansys Workbench

Ansys workbench is a solver which integrates other specialised solvers such as LS-DYNA and Autodyn, into single, streamlined project schematics. Advantages of this software include its ability to use multiphysics integration to connect structural analysis with fluids, as well as its ability to handle advanced materials robustly, even during high velocity impacts.

A disadvantage of this software is that it can have a steep learning curve for beginners when creating complex geometries and high quality meshes, where guidance may be required to understand.

It is used for ballistics impacts and aero-structural optimisation in the energy and aerospace sectors. [13, 14]

3.2 Abaqus

Abaqus is well understood for its ability to produce highly detailed results for large deformation problems. It allows seamless transition between implicit and explicit solvers with the same analysis, as well as detailed accuracy in simulating unpredictable contact and material failure.

Disadvantages of this software are the cost and compute resources, with licenses being expensive and the solver being memory intensive for large scale models.

It is used in the automotive industry primarily, for crash testing simulation as well as windscreen impact simulation in the automotive windscreen sector. It is also used in the metal stamping and forming industry. [13, 14, 15]

3.3 Fusion 360

Fusion 360 is a solver which offers event simulation as part of a cloud-based CAD/CAM/CAM system, providing quick validation with ease. It is cheaper and more intuitive than other Finite Element Analysis (FEA) packages, as well as being a cloud-based solver which offloads any heavy computation to Autodesk's own servers, allowing user to operate from low specification hardware.

Disadvantages of this software are its limited control over finite element aspects such as mesh refinement and advanced material laws; therefore, results are given in a basic environment and mesh detail.

It is used by smaller companies and startups for prototyping small-scale projects as well as for product drop testing [16, 17].

4 Conclusion

For this report, a brief introduction and history of CAD, CAM and CAE has been conducted, as well as a case study analysis of a projectile, a 9mm lead antimony bullet, impact on two plates of different materials, copper and 7039 aluminium, has been conducted to assess how CAE can be used as a tool for solving ballistic impact problems. This included modelling of the problem, meshing the model, setting the central coordinates and initial conditions and running the simulation to produce a series of solutions for deformation, stress, strain and velocity. Mesh convergence, mesh independence and boundary conditions were also studied to ensure any variables on efficiency and results were well understood. This report also contains validation of the results by comparison to other, similar studies conducted, with both simulated and real-world results compared against the case study to provide evidence to support the findings.

The findings demonstrated that aluminium provided a more brittle response to the projectile, however although copper is a softer material, it absorbed more of the kinetic energy of the projectile as the plate exhibited greater deformation, leading to the contact time between the plate and projectile also being greater and therefore slowing the velocity more than with the aluminium plate.

Through the validation it was understood that due to its softness and weight, copper is not often used as protection for ballistic impact, however it could be understood that if the copper is used as part of a composite material or structure, where weight is not an issue, then the kinetic energy absorption properties could be utilised. This was briefly touched on in sources but not fully specified or described.

Limitations of using solvers are that they cannot perfectly match any environmental conditions, nor replicate exact material properties and only make assumptions such as purity and/or distribution in the way of any anomalous impurities which may exist, particularly when using composite materials, the software distributes the composition evenly throughout. Other limitations are computing power, as explicit solvers require a lot of computing power to run

accurate simulations, which is why mesh independence was evaluated and analysed for, to ensure that excess computing power requirements which provided negligible benefits were not unnecessarily utilised.

Lastly a study and comparison between three solver software programs from three leading software companies was conducted, understanding the advantages and disadvantages of each, as well as their applications in industry. This was particularly insightful and could be used to understand the best way to solve any future problems which may be encountered as part of a professional research or safety study.

5 References

- [1] D. T. Ross, "Oral History Interview with Douglas T. Ross," *University Digital Conservancy*, Nov. 01, 1989. [Online]. Available: <https://conservancy.umn.edu/server/api/core/bitstreams/1a9bfedf-b991-46e6-bb10-fcce1f00056f/content> [Accessed: Apr 27, 2026].
- [2] P. Marks, "In remembrance of Dr. Jason A. Lemon, CAE pioneer," *GfxSpeak*, July 08, 2011. [Online]. Available: <https://gfxspeak.com/archives/in-remembrance-of-dr-jason-a-lemon-cae-pioneer/> [Accessed: Apr 27, 2026].
- [3] UCI Hall of Fame, "Patrick Hanratty," *Engineering and ICS: Hall of Fame*. [Online]. Available: <https://halloffame.tech.uci.edu/inductee/hanratty-patrick/> [Accessed: Apr 27, 2026].
- [4] I. E. Sutherland, "Sketchpad: A man-machine graphical communication system," University of Cambridge Computer Laboratory, Cambridge, UK, Tech. Rep. UCAM-CL-TR-574, Sept. 2003. [Online]. Available: <https://www.cl.cam.ac.uk/techreports/UCAM-CL-TR-574.pdf#:~:text=Ivan%20Sutherland's%20Sketchpad%20is%20one%20of%20the,recognized%20in%20his%20citation%20for%20the%20Tur%2D>. [Accessed: Apr 27, 2026].
- [5] LEAP Australia, "Tips & tricks: Convergence and mesh independence study," *LEAP Australia Blog*, May 24, 2017. [Online]. Available: <https://www.leapaust.com.au/blog/cfd/convergence-and-mesh-independent-study/> [Accessed May 1, 2026]
- [6] SimScale, "What are boundary conditions?" *SimScale Documentation*, Aug. 11, 2023. [Online]. Available: <https://www.simscale.com/docs/simwiki/numerics-background/what-are-boundary-conditions/> [Accessed May 1, 2026]
- [7] ASME, *An Illustration of the Concepts of Verification and Validation in Computational Solid Mechanics*, ASME V&V 10.1-2012 (R2022), 2012. [Online]. Available: <https://standards.globalspec.com/std/1519341/asme-v-v-10-1> [Accessed May 4, 2026]

- [8] M.A.I.Zakaria, M.N.Ibrahim. "Dynamic Response of AL2024 T3 Aluminium Plate Subjected to Oblique Impact of 9mm Bullet Using Single Processor" *RESEARCH PROGRESS IN MECHANICAL AND MANUFACTURING ENGINEERING*, Volume 1, Issue 1, 10 November 2020.
- [9] H. Mindivan, M. Baydogan, E. S. Kayali, and H. Cimenoglu, "Wear behaviour of 7039 aluminum alloy," *Mater. Charact.*, vol. 54, no. 3, pp. 263–269, Mar. 2005. [Online]. Available: https://www.researchgate.net/publication/248495730_Wear_behaviour_of_7039_aluminum_alloy [accessed May 5, 2026]
- [10] F. T. Teferi, K. P. Kolhe, A. A. Tsegaw, and O. S. Fatoba, "Elevating Al7039 alloy performance through copper addition for advanced industrial applications," *Results in Eng.*, vol. 26, p. 104823, June 2025. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S2590123025009004> [accessed May 5, 2026]
- [11] J.Hub, J.Komenda, M.Novák. "Ballistic Limit Evaluation for Impact of Pistol Projectile 9mm Luger on Aircraft Skin Metal Plate" *Advances in Military Technology*, Volume 7, Issue 1, June 2012.
- [12] H. J. Prentice, W. G. Proud, S. M. Walley, and J. E. Field, "Optical techniques for the investigation of the ballistic impact of thin plates," *Int. J. Impact Eng.*, vol. 38, no. 11, pp. 849–863, Nov. 2011. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S0734743X11001060> [Accessed May 4, 2026]
- [13] Engineering Downloads, "Abaqus vs ANSYS: Which simulation software is better in 2025?" *Engineering Downloads Blog*, Sept. 12, 2025. [Online]. Available: https://engineeringdownloads.com/abaqus-vs-ansys-comparison/#elementor-toc_heading-anchor-8 [accessed May 5, 2026]
- [14] D. S. Choudhary, "Abaqus vs Ansys for FEA: Key differences and use cases," *GaugeHow*, Jan. 6, 2026. [Online]. Available: <https://gaugehow.com/simulation/abaqus-vs-ansys-for-fea> [accessed May 5, 2026]

- [15] N. Shidid, "Abaqus solvers: Empowering engineers in every FEA scenario," *GoEngineer Blog*, Apr. 18, 2024. [Online]. Available: <https://www.goengineer.com/blog/abaqus-solvers-empowering-engineers-in-every-fea-scenario> [accessed May 5, 2026]
- [16] Top 3D Shop, "Modeling in Autodesk Fusion 360: In-depth review," *Top 3D Shop Blog*, July 24, 2022. [Online]. Available: <https://top3dshop.com/blog/modeling-in-autodesk-fusion-360-in-depth-review> [accessed May 5, 2026]
- [17] NetCom Learning, "AutoCAD vs Fusion 360: Comprehensive comparison in 2026," *NetCom Learning Blog*, Jan. 12, 2026. [Online]. Available: <https://www.netcomlearning.com/blog/autocad-vs-fusion-360-major-differences> [accessed May 5, 2026]