

Computer Aided Engineering – Bullet Impact FEA

Thomas Cresswell, 07 May 2026



Figure 1 – Cross-section of 9x19mm bullet [1]

Contents

Contents	2
List of Figures.....	2
List of Acronyms	4
Scope of assignment	4
1. Introduction to Computer Aided Engineering	5
2. Literature Review	6
2.1 Meshing.....	6
2.2. Assumptions	7
2.3. Boundary Conditions	7
3. Literature Validation Cases	9
4. Main Test Case	10
4.1. Variables of Interest	10
4.2. Methodology	10
4.2.1. CAD Modelling	10
4.2.2. FEA Software and Materials	10
4.2.3. Coordinate System	11
4.2.4. Simulation Conditions	11
4.2.5. Effect of Meshing on Deformation	11
4.2.6. Effect of Mesh Sizing on Stress/Strain Visualisation	14
4.2.7. Effect of Boundary Condition Application to Results	15
5. Technical Conclusions	17
5.1. Limitations of CAE	17
5.2. Accuracy Relative to Literature Validation	17
5.3. Impact of Key Assumptions	17
6. References	19

List of Figures

Figure 1 – Cross-section of 9x19mm bullet.....	1
Figure 2 – Comparison of a coarse and refined mesh on a plate [5].....	6
Figure 3 - Deformation results, low refinement mesh front	11
Figure 4 – Deformation results, low refinement mesh rear	12
Figure 5 - Deformation results, medium refinement mesh front	12
Figure 6 – Deformation results, medium refinement mesh rear	13
Figure 7 - Deformation results, high mesh refinement front	13

Figure 8 - Deformation results, high mesh refinement rear	14
Figure 9 - Elastic Strain results, low mesh refinement – 400 m/s	15
Figure 10 - Elastic Strain results, high mesh refinement - 400 m/s	15
Figure 11 - Elastic Strain results, medium mesh refinement - 600 m/s	15
Figure 12 - Elastic Strain results, high mesh refinement - 600 m/s	15
Figure 13 - Deformation results, high refinement mesh without boundary condition	16

List of Acronyms

CAD	Computer Aided Design
CAE	Computer Aided Engineering
FEA	Finite Element Analysis
JC	Johnson-Cook
SPH	Smooth Particle Hydrodynamics

Scope of assignment

This assignment aims to perform Finite Element Analysis (FEA) of a projectile impacting a plate across a range of velocities using a predefined test case. In addition, a literature review is conducted on the use of Computer Aided Engineering (CAE), as well as critical analysis of the key limitations and assumptions inherent within FEA.

1. Introduction to Computer Aided Engineering

Computer-Aided Engineering (CAE) is a subdiscipline of engineering that uses software tools to digitally simulate and optimize product designs. CAE is used across industries to aid in failure analysis, improve performance, decrease development costs, shorten design cycles, and give engineers insight into their product's performance [1].

CAE processes have become an essential part of the engineering cycle of products, allowing the derivation and partial validation of a design prior to the production of a physical prototype. The main benefit of CAE is the adaptability of the design whereby improvements and optimisations can be made in a much shorter timeframe than using physical representations of the design.

One of these processes within CAE is called Finite Element Analysis (FEA). This process is the simulation of stimuli applied to a computer model of the prototype. Within this assignment, the CAE tools used are ANSYS, an FEA simulation suite, and Fusion 360, a 3D Computer Aided Design (CAD) tool. FEA works by building a mathematical representation, consisting of nodes, of how the model's design and materials are influenced by predetermined loads. This simulation uses a complex set of calculations repeatedly in order to determine this influence.

The Johnson-Cook constitutive model was derived in 1985 by Gordon R Johnson and William H Cook. This model is a constitutive flow-stress model which expresses stress as a function of strain, strain rate, and temperature[2].

The mathematical expression of the model is $\sigma = (A + B\varepsilon^n)(1 + C \ln \dot{\varepsilon})(1 - T^m)$ where [3]:

- σ : Flow stress
- ε : Equivalent plastic strain
- $\dot{\varepsilon}^* = \dot{\varepsilon}/\dot{\varepsilon}_0$ is the dimensionless plastic strain rate normalized by a reference strain rate.
- $(A + B\varepsilon^n)$: Represents strain hardening where A is the yield stress, and B and n are the hardening modulus and exponent.
- $(1 + C \ln \dot{\varepsilon})$: captures strain rate effects through constant C.
- $(1 - T^m)$: Accounts for thermal softening, where $T^* = (T - T_r)/(T_m - T_r)$ is the temperature normalized between reference temperature T_r and melting temperature T_m , with m as the thermal softening exponent.

The way that each of these constants (A, B, n, C, m) are determined is a sequential calibration process utilising experimental data from testing conducted at reference conditions. This allows for isolation of each of the term's contribution, isolating the effects of strain, strain rate and temperature on the flow stress.

2. Literature Review

This section contains a technical review of accepted and known limitations of the FEA software as well as the calculations and principles which surround it. The review is focused on simplifications and assumptions in non-linear scenarios and how they influence the accuracy and validity of results. The effects of boundary conditions and their impact on the simulation will also be discussed.

2.1 Meshing

Meshing is the process of breaking up a larger model of a part into a set number of smaller geometric shapes called elements. The elements are connected at points called nodes. During the calculation of a solve, the number of nodes within the simulation governs how much processing and therefore time is required to complete the solve, the greater the number of nodes the greater the time taken to solve. When using FEA as a cost/time-saving tool, this principle becomes important.

One of the main considerations for meshing is the density of the mesh. This is the number of elements the model has within its volume. A higher mesh density will increase the accuracy up to a point of convergence. This is due to the smaller elements better representing the deformation. A journal article published to Materials (Basel) discussed the influence of mesh density on the results obtained by FEA [4]. Within this journal, the solution accuracy improved with the introduction of a finer mesh. The tested range was approximately 1000 – 100,429 elements, increasing the mesh density with each iteration per example. The deviation of the solution compared to real life testing tended to zero towards the 100,000-element example across all of the simulations. The largest deviation showed up to a 29.4% deviation in deformation shown, giving example as to the dangers of a coarse mesh. When refining the mesh, there are two types of refinement. H-Refinement is the process of reducing the element size. This is useful where there are stress gradients, complexity of the geometry and zones of contact [5]. P-Refinement is where the mesh style is changed to one with more nodes per element. The use case of P-Refinement is where there is a lower mesh density or an excess in processing capacity for the solution.

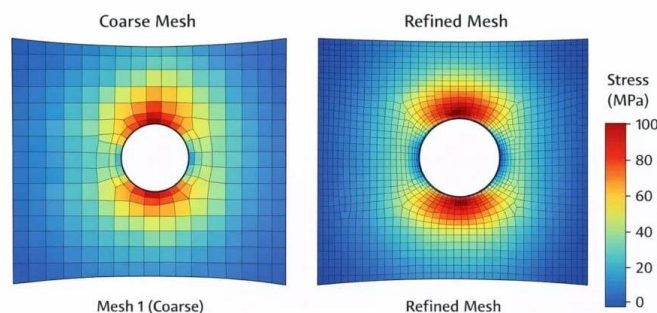


Figure 2 – Comparison of a coarse and refined mesh on a plate [5]

Determining the correct meshing type/style is also important when producing an FEA simulation. The types of meshing available within ANSYS were Hex, Tet, Tri and Quad. Whilst there is no universal best mesh type, the type of mesh will have impacts on the results and time taken to process which may not be optimal for the task at hand.

The choice of style will fundamentally affect the precision of the solution and quad style elements will usually produce a more accurate result in a 2D environment since linear quads can suffer from shear locking. When dealing with structural style problems, triangle elements can cause artificial stiffness unless the mesh sizing is small which may impact solves using a coarser mesh for efficiency. This is due to the element shape distributing stress differently. Hex meshes have benefits within contact and thin-walled parts within the solution due to the shape reducing rounding errors. The aspect ratio of elements is recommended to remain at a maximum of 5:1 since greater values can reduce the accuracy of the solution.

2.2. Assumptions

Materials used within FEA are mathematically idealised and have two common assumptions, that the material is homogenous and isotropic. Isotropic materials are assumed to have identical properties in all directions. This can have inaccuracies when dealing with rolled metal sheets and composite materials. Homogenous materials have the same material properties throughout the whole material; this is not completely accurate for casted metals due to porosity or heat-treated surfaces with a hardened surface.

The Johnson-Cook constitutive model has multiple assumptions within the equation. The model assumes that the strain hardening, strain rate hardening and thermal softening of the materials are multiplicative. With 6061-T6 aluminium, the n exponent is low which is the work hardening modulus. This means that necking will occur rapidly and the model is likely to overpredict the ductility of the material. These overpredictions could lead to overpenetration of the armour plates used within the example.

When dealing with a high-speed ballistic impact in FEA, element sizing plays a large role in the accuracy of the resultant simulation. A ballistic impact is heavily dominated by material fracture and adiabatic shear; this will lead to larger elements naturally resisting failure reducing the accuracy of the simulation.

During the simulation, some elements which fracture away will be deleted once they reach a predetermined threshold of damage. This is a convenience for the software as it reduces the processing time for the simulation since usually these are individual nodes. This introduces multiple artefacts to the results since fracture is a progressive process whereby breakaway pieces still interact with the remaining material. A finer mesh will reduce the element deletion since the criteria for element deletion will be altered.

The density of the mesh used on the model has a direct impact on the computation time of the simulation through its impact on the stable time increment used by the simulation. This time step is determined by the smallest element that is present on the model.

2.3. Boundary Conditions

A boundary condition within FEA is a constraint necessary for the solution of a boundary value problem [6]. This is where there is a differential equation to be solved, in the case of FEA Johnson-Cook constitutive equation, where the boundary is a set of known conditions. Within

FEA, the control of boundary conditions and choice of meshing style can have a significant impact on the results and accuracy of the simulation.

Boundary conditions can be used for a multitude of functions within FEA. Symmetry boundary conditions allow the selection of a series of edges or faces where the solver will simulate one half and then provide a visual representation across the selected axis. Boundary conditions are also used for representing supports and fixities through the use of fixed supports, elastic supports or roller supports. These allow the model to be held in a way that is representative of its use ensuring the stresses and deformation are more accurately represented. Properly defined boundary conditions will enable convergence of the solution by reducing the variability of the scenario. The increased stability of the scenario also allows for a more in-depth analysis of the other variables within the solution since they become more isolated.

3. Literature Validation Cases

This section details the significance of identifying an appropriate test case and its use in validation of the software for the intended task. A literature-based validation is discussed and its relation and findings are explained in the context of the main test case.

When performing FEA on a terminal ballistics test case, it is a highly complex simulation with high stress occurring within a very small window. As a result of this, it is important to verify the use of the software, selected materials and mesh sizing. This is because the sources of error are easily obscured within uncertainties in the material models and contact behaviour. This validation can be performed using a simplified test which is representative of the load as well as real life testing of the example intended to be simulated. In linear range examples, like a beam bend, hand calculations can be performed as a deterministic benchmark for the simulation.

The high-carbon steel ball bearing impacting an aluminium plate was chosen as the simplified test case due to its presence in the literature, where multiple institutions have documented experimental results that provide a robust basis for comparison. In this test case, the uncertainty of the expected results is reduced since they are both hard materials and it is an example which has been tested using gas guns.

A mesh convergence study is recommended when setting up a scenario for FEA. This is where a series of solves are conducted using a selected test case, gradually refining the mesh of the components until a point of convergence is met in the stress and deformation. The point at which the results converge to an acceptable tolerance is called the point of mesh independence.

A journal article on High-Velocity Ball Penetration of D16 Aluminium is a real-world example of this test case [6]. Within these experiments the researchers utilised a powder accelerator to launch the ball bearings across a velocity range of 600-1000m/s into the plate. An FEA analysis performed in LS-DYNA was developed as a result of these experiments. The aluminium alloy used within this example is an aluminium, copper and magnesium composition which contains similar properties to 6061-T6. Within this testing, the solution was able to be refined to a convergence within 10% of the tested deformation values.

Another validation paper on a 7.62 mm projectile into an Aluminium 6061-T6 plate displays the real-world behaviour of the interaction [8]. Within the report, it concluded that the refinement of their mesh improved the qualitative accuracy of the result displaying more accurate 'petaling' and deformation. An observation was made however than in the 0-degree oblique impact, inaccurate values for the categorisation of their brass coating may have increased the strain at failure by up to 40%. This highlights the importance of the use of validated values for materials within the simulation.

4. Main Test Case

This section covers the critical evaluation and assessment of qualitative and quantitative data generated by an FEA simulation, the setup of the solution and the generation of the models for the parts.

4.1. Variables of Interest

The variables of interest within this test case include:

- Deformation of the plate and projectile – A qualitative assessment of the deformation pattern as well as maximum deformation value.
- Velocity over time for the projectile – Visualised through graphing and record of the minimum velocity of the projectile, remaining velocity is confirmation of a penetration of the plate.
- Stress and strain – The maximum principal stresses and strain for the plate

4.2. Methodology

Since the literature validation of the test was based on a curved surface impacting a hard plate, the chosen scenario was a 9mm projectile hitting a 1.5mm thick armour plate at increasing velocities. The variables of interest within this simulation are; the deformation of the plate, maximum principal stresses, maximum principal shear stresses and velocity of the projectile over time. This will allow for a qualitative assessment of the failure mode of the plate and the non-linear nature of the scenario will display some of the principles discussed within section 2 of this report.

4.2.1. CAD Modelling

To generate the models for the test, Fusion 360 was used to create a 3D model of the projectile and the plate. Fusion 360 is a Computer Aided Design (CAD) package which is used within CAE to produce 3D models and engineering drawings of parts. For the projectile, the dimensions of a Luger branded 9mm projectile were referenced and for accuracy an ogive and boattail were included in the model. Once the 3D models were produced, they were imported into ANSYS Workbench where all of the models, engineering data and simulation information is stored within the chosen package.

4.2.2. FEA Software and Materials

The chosen FEA software is ANSYS Explicit Dynamics, a solver which is intended for use in high-speed interactions between components. The software calculates the problem through the use of time sets where the deformation and stress waves are tracked. A well-known concept within CAE is 'Garbage-in-Garbage-out', this is the idea that inaccurate data, measurements or properties will lead to an unusable and inaccurate result for the simulation. When choosing the materials for use within the solution, ANSYS Granta was used as a library of verified sources for material data which is integrated with Workbench. It is important that when dealing with an explicit solve, the information collected from Granta is from the explicit

section of the library. Key information, like the Johnson-Cook properties of a material may be missing from the linear version of the material data. In this case Lead Antimony 96-4 Alloy and 6061-T6 Aluminium were selected as appropriate materials for the test case.

4.2.3. Coordinate System

A coordinate system is present within the solver which define the reference frame for the meshes present and the loads that will be applied to those meshes. The loads applied within the test case are 2 different initial velocities; 400 m/s and 600 m/s for the projectile. This simplified the measurement of displacement of the plate and the velocity of the projectile over time.

4.2.4. Simulation Conditions

Once the meshes had been applied and are aligned with one another, conditions are required to be set for them in order to achieve an interaction within the solve. A velocity load is applied to the tip of the projectile under the Initial Conditions section of the software ensuring the vector is aligned with a 0-degree oblique to the plate. In most of the test cases, a boundary condition was applied to the edge of the plate to fix it in place. This prevents a loss of energy from the projectile from the movement of the plate absorbing partially the impact as the plate gains velocity.

4.2.5. Effect of Meshing on Deformation

To determine the effect of meshing within the selected test case, 3 different levels of mesh refinement for both the projectile and the plate were used and will be referred to as low, medium and high.

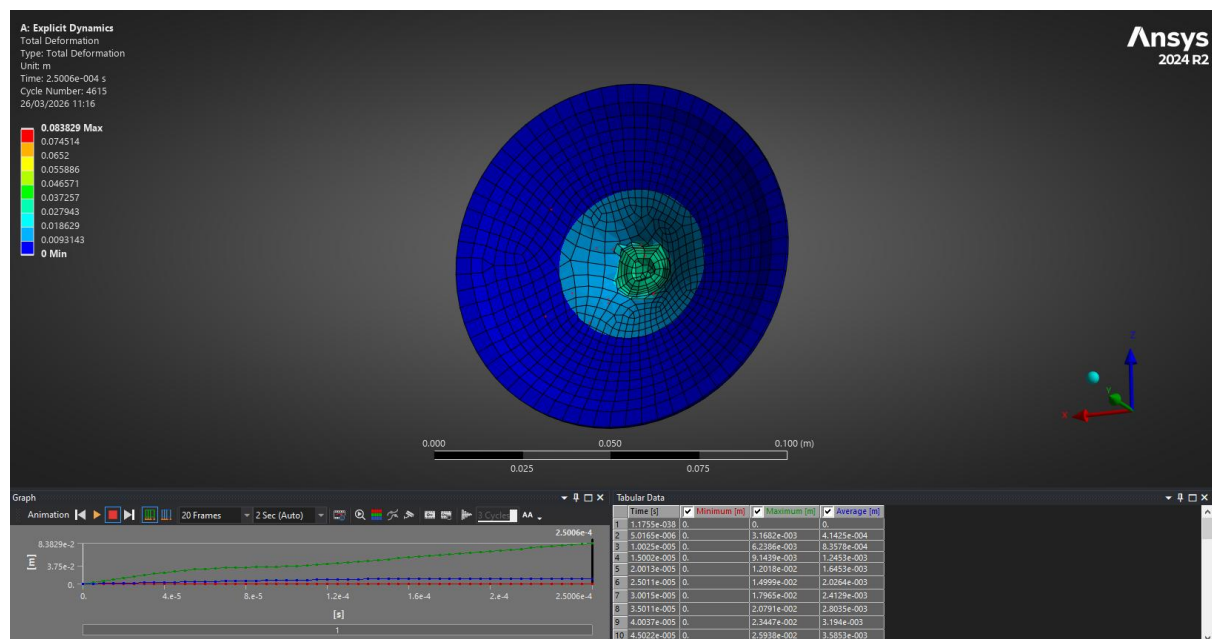


Figure 3 - Deformation results, low refinement mesh front

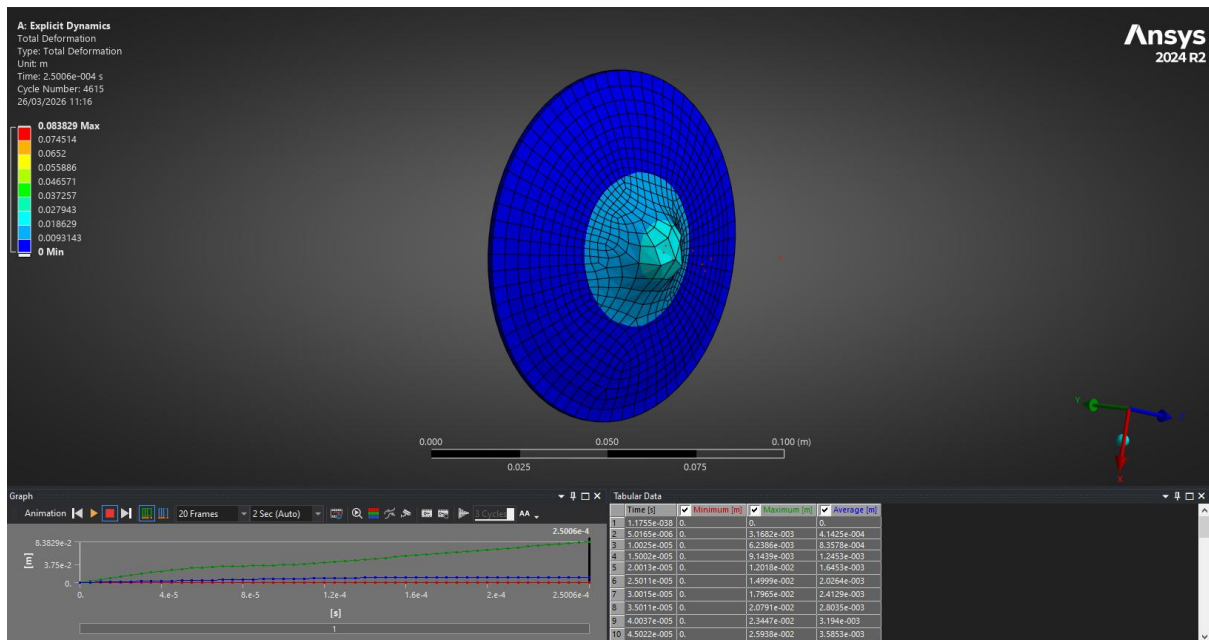


Figure 4 – Deformation results, low refinement mesh rear

As per figures 3-4, the projectile fails to penetrate the plate leaving a coarse bell-like deformation pattern. The deformation of the plate in this instance was measured at approximately 0.028m.

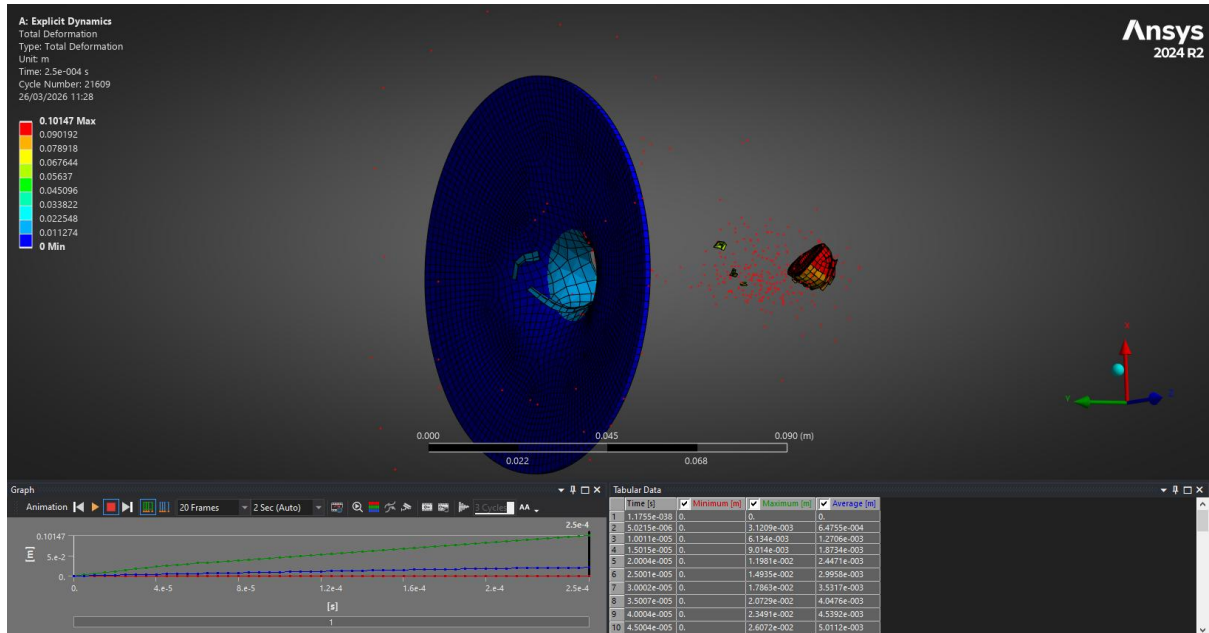


Figure 5 - Deformation results, medium refinement mesh front

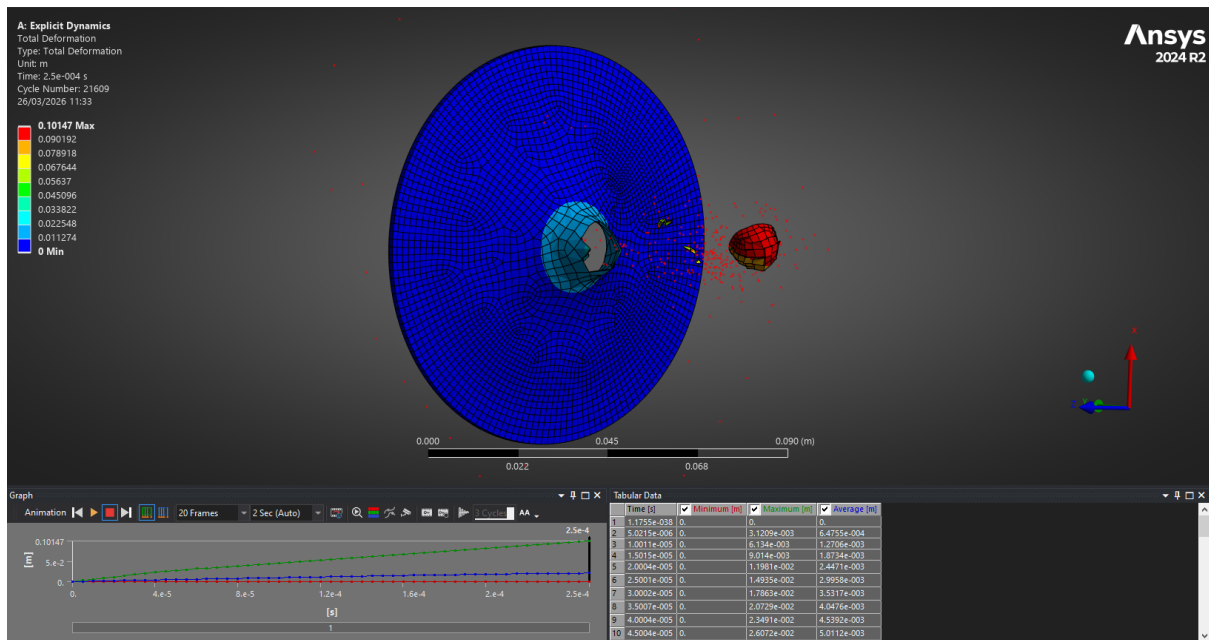


Figure 6 – Deformation results, medium refinement mesh rear

As seen in Figures 5-6, the deformation of the refined plate mesh resulted in a penetration of the plate in comparison to a non-penetration with the coarser mesh. This is likely due to the element erosion thresholds being different based on element size. The result is a coarser mesh is unable to accumulate enough 'damage' to be fractured or deleted. The maximum deformation of the plate is now recorded at approximately 0.034m.

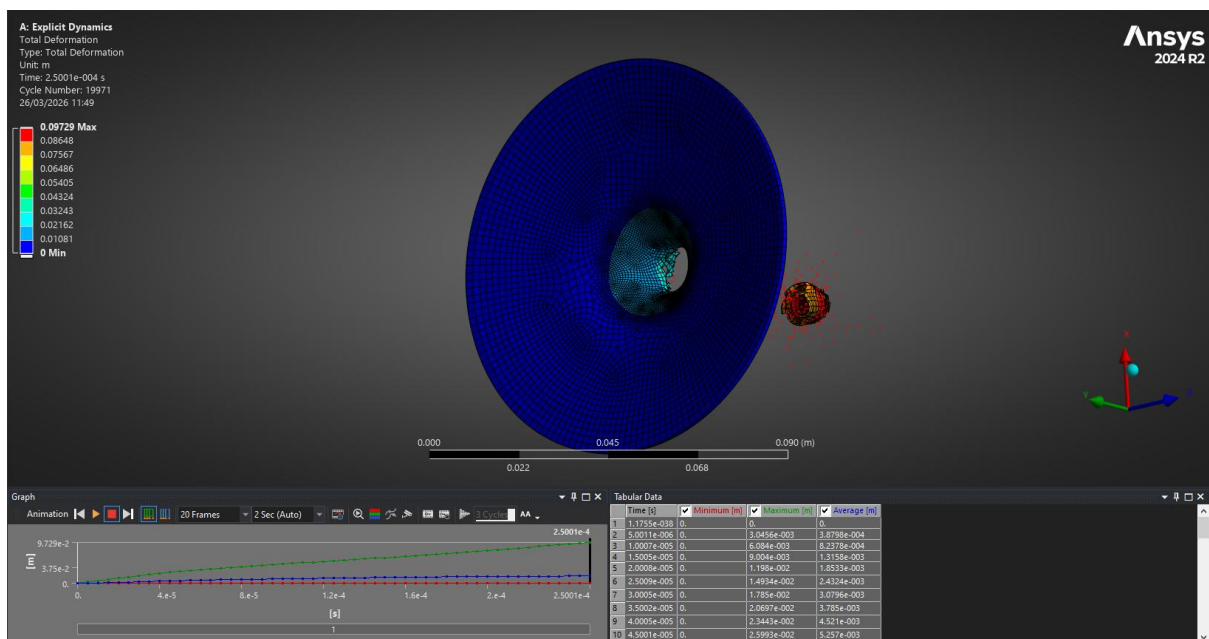


Figure 7 - Deformation results, high mesh refinement front

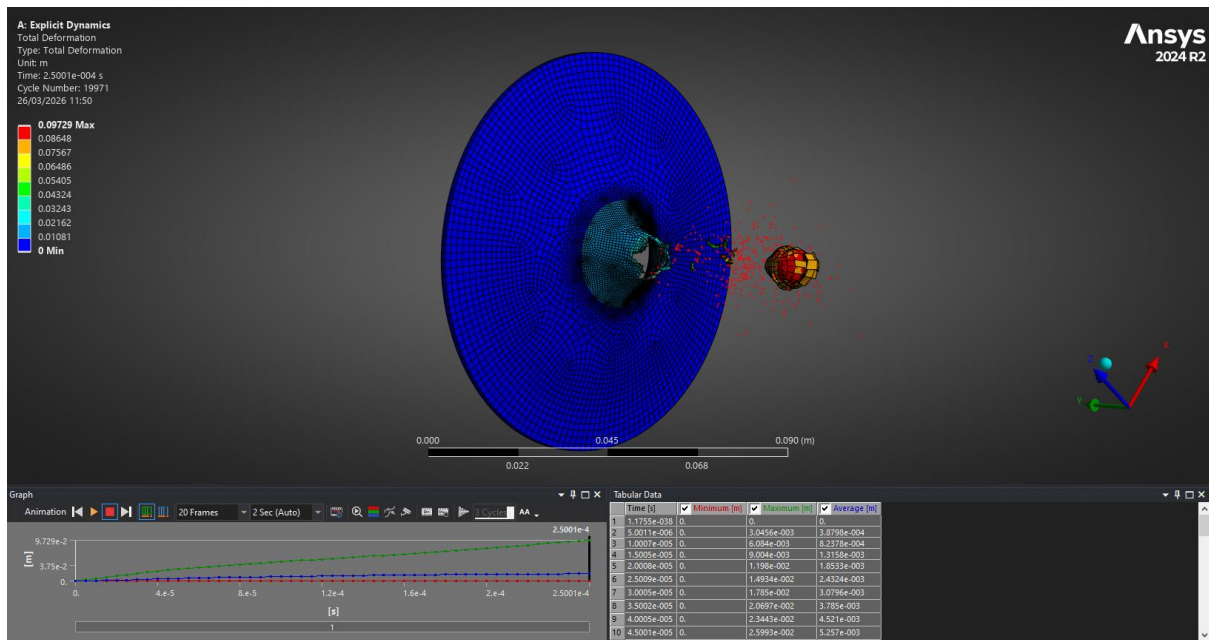


Figure 8 - Deformation results, high mesh refinement rear

Figures 7-8 detail the reduction of element deletion with finer meshes as spall elements become more visible. This is due to the minimum element size being reduced alongside an increase in nodes allowing for small pieces of material to be simulated fracturing from the main piece. This provides further qualitative information surrounding the after-hit effect of the projectile which can influence design decisions. The deformation displayed in these figures is a maximum deformation of the plate of 0.032m, this value is potentially affected by the spalled elements. This puts this estimation within 7% of the medium meshes value suggesting that for the purpose of efficient estimations of effect, the medium mesh could be suitable if detailed spall is not a consideration of the design. Spalling is fragmentation ejected from a rear surface.

This set of testing demonstrates the importance of mesh refinement within FEA whereby increasing the refinement of the mesh will provide an increase in qualitative data as well as a more consistent quantitative result.

4.2.6. Effect of Mesh Sizing on Stress/Strain Visualisation

Since identifying stress and strain within the parts/materials is vital to the analysis of a design, this section explores the effect of mesh sizing on the display of this information as well as its effect on the values.

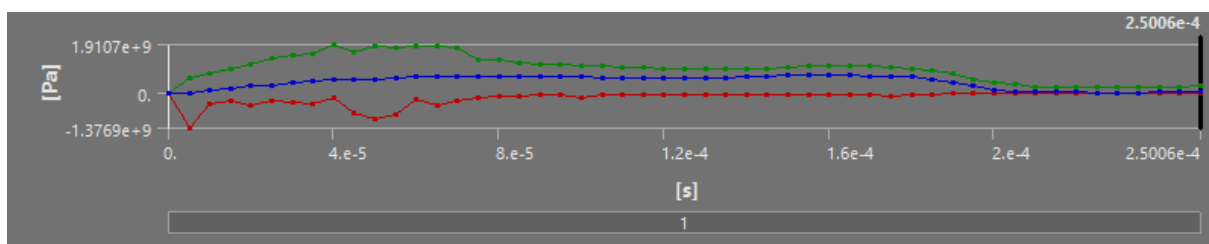
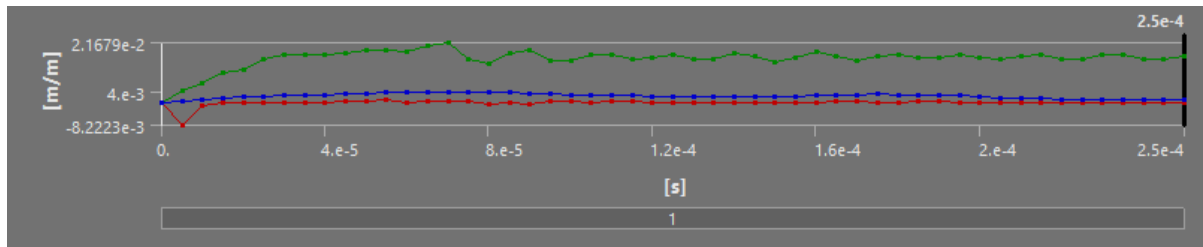
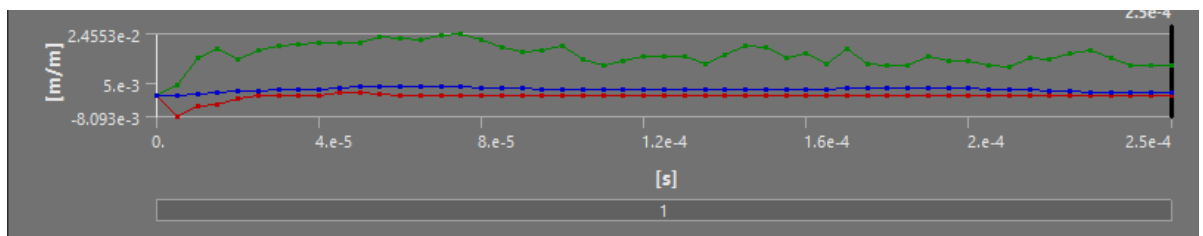
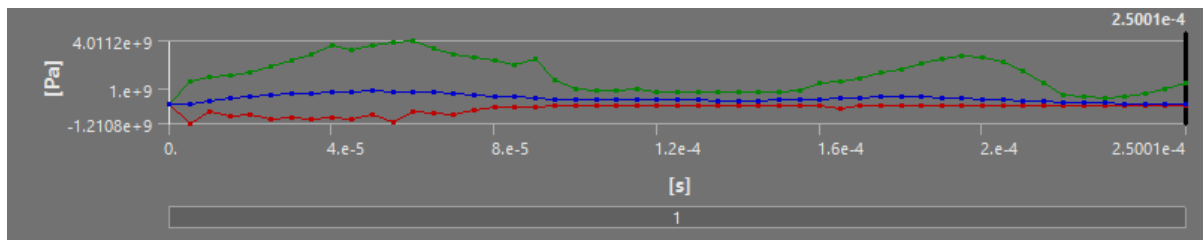


Figure 9 - Elastic Strain results, low mesh refinement – 400 m/s**Figure 10 - Elastic Strain results, high mesh refinement - 400 m/s**

As shown within Figures 9-10, the strain values also differ significantly with mesh refinement. The strain peaks at a similar time however the higher refinement displays that the strain has maintained its level whereas the coarser mesh loses strain over time.

**Figure 11 - Elastic Strain results, medium mesh refinement - 600 m/s****Figure 12 - Elastic Strain results, high mesh refinement - 600 m/s**

Figures 11-12 detail the same pattern experienced in Figures 9-10 whereby the strain values differ in pattern and in intensity. When using FEA within CAE to determine the appropriate designs, an accurate set of values for the stresses and strain which can be tolerated by the part is essential for validating the design.

4.2.7. Effect of Boundary Condition Application to Results

By analysing the stress and strain information provided within the graphs and coloured gradient within the software, the likely mode of failure can be determined for the plate. Since the scenario is a ballistic impact, adiabatic shear would be the expected failure mode for a penetration of the plate.

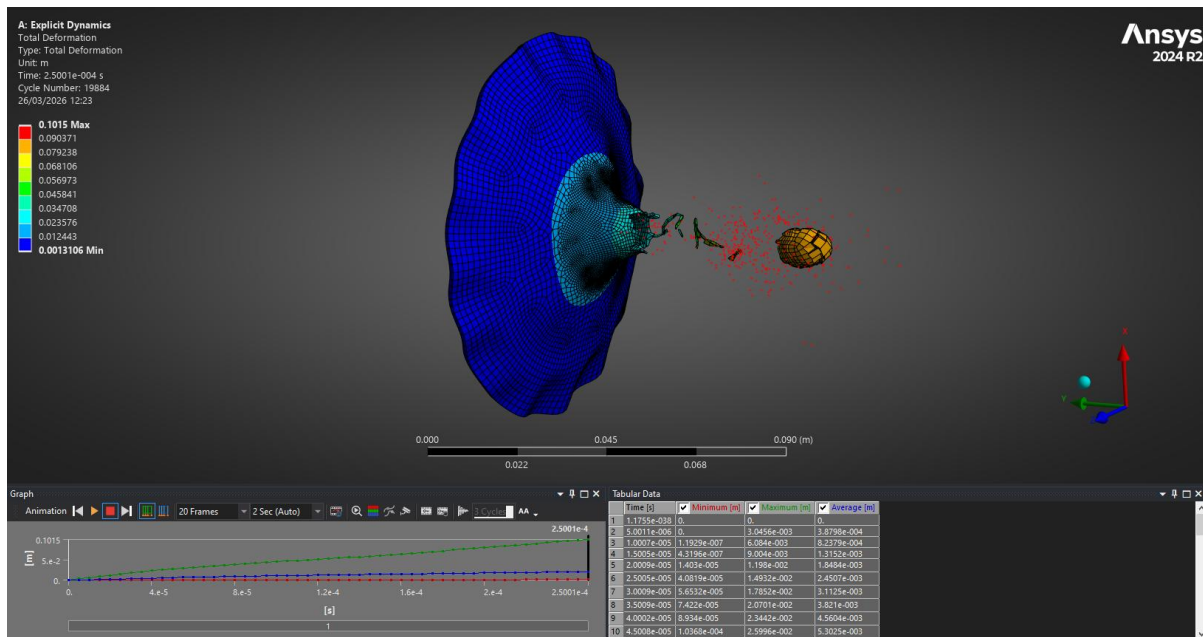


Figure 13 - Deformation results, high refinement mesh without boundary condition

As per Figure 13, the removal of the boundary condition resulted in a visible transfer of energy to the edges of the plate forming a pleating effect. This instance did not significantly affect the deformation however with a maximum deformation of the plate being 0.0347m which is still within 10% of the clamped, high refinement mesh. In the case of a symmetry boundary, the simulation time of the test case in practice could be shortened to provide more justification for the use of the high refinement mesh.

5. Technical Conclusions

The technical conclusions have been separated into sections to highlight each of the important areas within the scope of the assignment. This section provides each conclusion and summarises the key findings from the activities.

5.1. Limitations of CAE

Both the literature review and the ANSYS simulation work displayed the significance of mesh sensitivity with respect to the quality of the data produced by the solver. The coarse mesh failed to predict the penetration of the plate at whereas the medium and high refinement meshes successfully predicted penetration of the plate at 600 m/s. The maximum difference in deformation was 21%, where the coarse mesh predicted 0.028m and the high mesh predicted 0.034m of deformation with a penetration. This variance in the deformation directly displays the influence of the artificial resistance to damage larger element sizes have within FEA activities and the significance of the erosion threshold present within the calculation.

Element deletion also introduced artefacts as displayed between the medium and the high refinement meshes. Only the high refinement mesh displayed clearly the spalling and discard of material from the plate shearing. This is a fundamental flaw in the Lagrange method of FEA whereby the element erosion model cannot reliably predict the spall or after effect of a munition once it is through the armour plate.

5.2. Accuracy Relative to Literature Validation

Within the literature validation, Anisimov and Soliman had achieved a value within 10% of experimented deformation values for their steel ball impact on the aluminium plate. The plate had a similar material and the pattern of damage from a penetrating projectile qualitatively matched. The simulations ran within the main test case remained within 7% of one another, slowly converging into a more stable result. This suggests that the main test case had successfully moved towards mesh convergence.

Due to no experimental testing, the main test case cannot be completely validated. Within the validation papers, a difference in the projectile is common (steel ball bearing instead of 9mm). The simulations ran within the main test case can be considered to be qualitatively comparable and will accurately display the failure mode and effects on the material. However, direct values for stresses and penetration limits would require direct experimentation as a proof.

5.3. Impact of Key Assumptions

Within the main test case, the impact of the assumptions made within the software is noticeable. The homogenous and isotropic material properties would not be displaying the likely surfaced hardened plate from being rolled during manufacture. The effects of material grain orientation and any residual stresses from manufacture would also have a slight impact on the true results of this scenario.

The assumptions made by the Johnson-Cook model within the calculations will also have impacted the result found. This is thorough the typically discussed overprediction of ductility and therefore penetration of the materials. This is from the low n exponent within the materials used and cannot be used for armour design since it would lead to a non-conservative size and depth of plate where it is not needed. In contrary, the design of the projectile would be negatively affected by displaying an overly optimistic view of penetration whereby the projectile may need a change in design if being made for a pre-defined threat.

Finally, the element erosion within the Lagrange method of FEA impacted the display of discarded material within the scenario. Typically, slight material discard will be present on the wall of the penetration as well as spall following the exit vector of the projectile. This spalling is incorrectly displayed since finer elements are deleted. For accurate spall prediction, a witness plate should be installed during experimentation.

6. References

- [1] J. Kottke, "Bullet cross-sections," [Online]. Available: <https://kottke.org/13/06/bullet-cross-sections>. [Accessed 07 05 2026].
- [2] ANSYS, "What is Computer-Aided Engineering (CAE)?," 2026. [Online]. Available: <https://www.ansys.com/simulation-topics/what-is-computer-aided-engineering>.
- [3] G. R. Johnson and W. H. Cook, "FRACTURE CHARACTERISTICS OF THREE METALS SUBJECTED TO VARIOUS STRAINS, STRAIN RATES, TEMPERATURES AND PRESSURES," *Engineering Fracture Mechanics*, vol. 21, no. 1, pp. 31-48, 1985.
- [4] G. Kharmanda, H. Mulki, S. Gowid and A. Shokry, "On the Prediction of the Flow Behavior of Metals and Alloys at a Wide Range of Temperatures and Strain Rates Using Johnson–Cook and Modified Johnson–Cook-Based Models: A Review," *Materials*, vol. 16, no. 4, p. 1574, 2023.
- [5] C. Pisarcuic, I. Dan and R. Cioara, "The Influence of Mesh Density on the Results Obtained by Finite Element Analysis of Complex Bodies," *Materials (Basel)*, vol. 16, no. 7, 2023.
- [6] P. SANAP, "Mesh Sensitivity, Convergence, and Accuracy: What Makes FEA Results Trustworthy," Ideametrics, [Online]. Available: <https://ideametricsglobalengineering.com/convergence-in-fea-analysis-validation-guide/>. [Accessed 04 05 2026].
- [7] "What Are Boundary Conditions?," Simscale, [Online]. Available: <https://www.simscale.com/docs/simwiki/numerics-background/what-are-boundary-conditions/>. [Accessed 2026 05 04].
- [8] A. G. Anisimov and M. E. Ahmed Soliman, "Simulation of the Impact of a Steel Ball on an Aluminum Plate D16 and Comparison with the Experiment," *Journal of Machinery Manufacture and Reliability*, vol. 53, no. 1, pp. 66-72, 2024.
- [9] M. Giglio, A. Gilioli, A. Manes, L. Peroni and M. Scapin, "Investigation about the influence of the mechanical properties of lead," *EPJ Web of Conferences*, vol. 26, no. 1, p. 04010, 2012.