

20240309 - CAE Coursework CTH

by Charles Hadley

Submission date: 07-May-2024 09:08PM (UTC+0100)

Submission ID: 232611440

File name: 42315_Charles_Hadley_20240309_-_CAE_Coursework_CTH_569859_1646906504.pdf (1.56M)

Word count: 3541

Character count: 20143



Charles Hadley
DE&S OME Degree Student

+44 7919977349

Charles.Hadley104@mod.gov.uk

UWTSD IQ Campus,
Waterfront,
University of Wales Trinity,
Swansea
SA1 8EW

Charles Hadley

7-May-24

Our Reference: CAE CW_01

EXPLORING A BALLISTIC PROBLEM WITH COMPUTATIONAL MODELING

References. See footnotes and references section 13.

1. Abstract

The intent of this report is to evaluate Ansys Workbench for a ballistic problem with comparison to Abaqus. To this effect it details a credible scenario on a bullet impacting a target plate. It will be exploring when and how the plate fails as well as different aspects of modelling techniques. It will detail assumptions and limitations of the simulation and how these can affect Computer Aided Engineering (CAE).

2. Table of Contents

1. Abstract	1
2. Table of Contents.....	2
3. Introduction and Background	3
4. Geometry	4
5. Meshing:	5
6. Materials	6
7. Validation	7
7.1. Material Validation	8
8. LS dyna vs Explicit Dynamics	11
9. Setup	11
10. Results	12
10.1. Ansys Results.....	12
10.2. Abaqus Results [7]	12
10.3. Comparison	12
11. Limitations and Assumptions	13
12. Conclusion	13
13. References	14
14. Annex A.....	15
15. Annex B.....	16
16. 300mps Ansys Explicit Dynamics Side Cross Section view	16
17. Annex C: Abaqus Model 2 Results	19

3. Introduction and Background

This report will explore a ballistic impact model via Finite Element Analysis (FEA) using Ansys Workbench. The aim of this report is to demonstrate how a given bullet impacts a target plate in relation to principle and shear stress. These results will then be used to compare to previous studies¹ using Abaqus to best evaluate the limitations of Computed Aided Engineering (CAE).

The chosen scenario that this report will consider is the impact of a subsonic 7.62mm x 39mm round on aluminium plate. This will be modelled with a representative round impacting an aluminium plate. The aluminium plate is assumed to be representative of a lightweight armour. The data achieved from models such as the ones in this report could be used to evaluate the effectiveness of protective structure used on many vehicles and structures. [1]

This scenario is a likely event as the 7.62mm x 39mm round is a popular calibre in ex-soviet states. This is due to the availability of AK-47's and their ammunition is widespread. Therefore, this presents a likely scenario of a bullet impacting an aluminium armour plate. [1]

When a steel cored bullet impacts hard armour little to no penetration is provided by the soft metal jacket. This jacket is simply left splattered on the outside of the plate as it is scraped off the core. The jacket simply provides a more malleable material for the bullet to 'grab' the rifling of the weapon upon firing. In this scenario it is assumed that the copper jacket would not provide a meaningful increase in the penetration and can therefore be excluded. This will simplify the simulation by removing the copper jacket from the model and only using the core impacting the armour plate.

¹ As this report will closely compare previously submitted work some areas will bear close resemblance, this has been referenced where appropriate [1]

4. Geometry

The bullet geometry used within this scenario has been simplified from a similar paper using a 7.62mm AP round [2].

The models were created in SolidWorks using a uniform sketch between the two parts and a simple revolve, these can be found below in Figure 1.

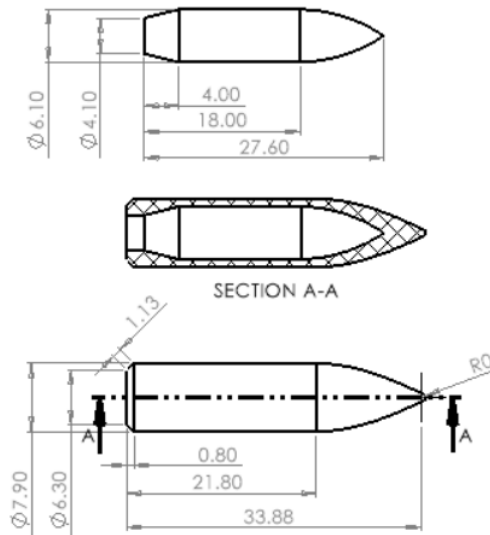


Figure 1 – 7.62 mm x 39mm Bullet Geometry

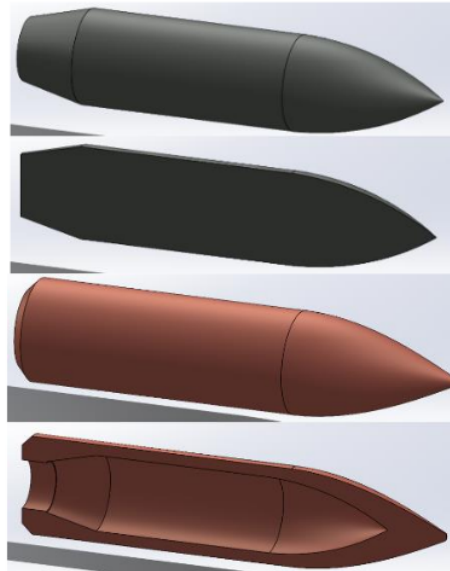


Figure 2 - 7.62 mm x 39mm Bullet Model

The target plate geometry used in this simulation was simplified to be a circular puck with 3 split regions to allow for more effective meshing this is shown in Figure 3. This representative target will allow for faster simulation due to the reduction of nodes. The size of the plate was determined from practical understanding of bullets impacting a plate.

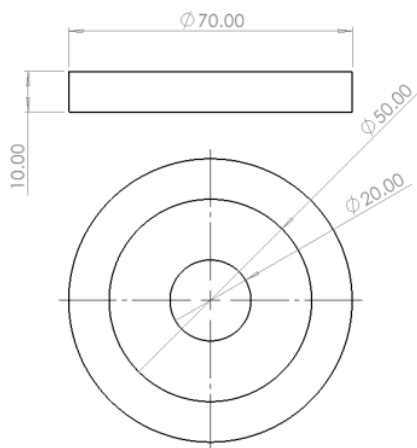


Figure 3 – Target Plate Geometry

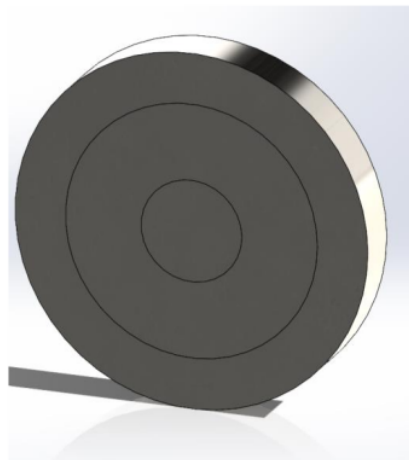


Figure 4 – Target Plate Model

5. Meshing:

Through trial and error, a suitable mesh was determined to best balance simulation time and sufficient nodes for more accurate results. In addition, the size of the nodes at the centre of the target is like that of the projectile as a large miss match in node size can cause issues in the simulation. By slowly increasing the number of nodes towards the centre of the target greater detail can be achieved where the impact will take place. This will reduce simulation time as a large number of nodes at the outer of the plate would be unnecessary and waste compute time. This balance of node size should provide sufficient information to the simulation without being a burden.

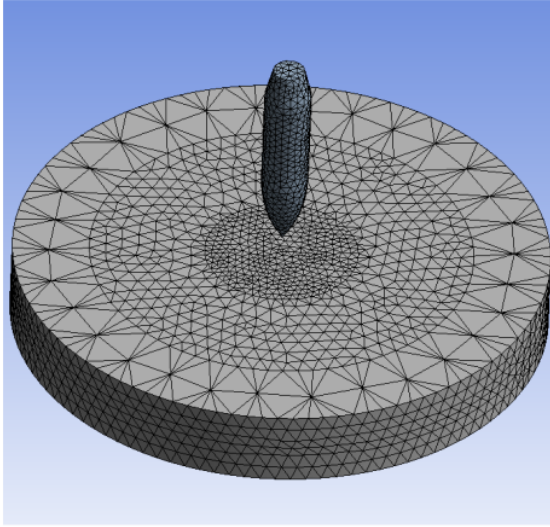


Figure 5 – Model Mesh

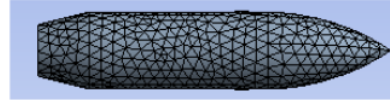


Figure 6 – Bullet Core Mesh

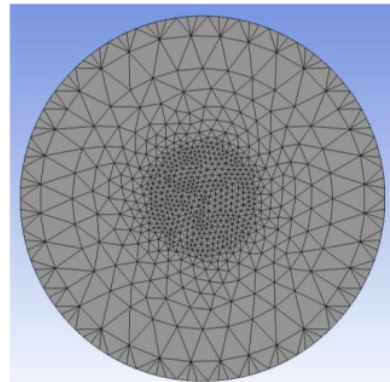


Figure 7 – Target Mesh

Face / Location	Node Size (mm)
Projectile	0.001
Target Outer Ring	0.001
Target Middle Ring	0.002
Target Inner Ring	0.005
Target Edge	0.002

Table 1 – Mesh Node Sizes

6. Materials

For this scenario the materials chosen were aluminium for the armour plate as this is a common material for lightweight armour and steel for the core of the bullet. The specific material properties used are detailed below. These properties include Johnson-cook values for strength to provide more accurate data for the simulation to use.

Material	Steel S-7 [3] [4]	AL 5083H116 [5]
Density (kgm^{-3})	7750	2700
Young's Modulus (Pa)	70×10^9	70×10^9
Poisson's Ratio		0.3
Yield stress A (Pa)	1.539×10^9	1.67×10^8
Hardening constant B (Pa)	4.77×10^8	5.9610^8
Strain rate constant C	0.012	0.001
Thermal softening exponent M	1	0.859
Hardening exponent N	0.18	0.551
Melting Temp TM ($^{\circ}k$)	1489.9	893
Transition Temp ($^{\circ}k$)		300
Failure Parameters		
D1	-0.1895	0.0261
D2	0.7324	0.263
D3	0.6633	-0.349
D4	0.0291	0.247
D5	0.7162	16.8
Reference Strain Rate	1	1
Shear Modulus	8.18×10^{10}	2.69×10^{10}

Table 2 – Materials Data

The above properties were drawn directly from the available explicit materials library in Ansys but were compared to published and known results [5] [3] [4] to ensure accuracy and validity. In an ideal scenario custom material data would be developed using real world testing and calculations to achieve more accurate results but for the scope of this exercise using existing data was the most suitable option.

7. Validation

Validation is a critical step in all simulation activities. It allows the real world to be compared to the digital by either calculating expected values and then comparing these to digitally generated results or by performing real world tests and then performing the same test digitally to understand the differences.

There are many approaches that can be used to validate simulation models, for example a small-scale model could be put in a wind tunnel to compare against a scale computational fluid dynamics assessment. To understand if the results are similar prior to full scale testing.

Another method that can be used would be a simple beam bending model as this scenario are greatly understood and could be calculated by hand and compared to the simulation.

As with any simulation there are always going to be differences but if these differences can be repeated and standardised that can be accounted for and understood in the model further improving accuracy.

In the case of this exercise a simple mechanical bending or static force validation cannot be used due to the complexity of the scenario. Instead, the chosen solver and materials can be validated using a tensile strength test. A quasistatic tensile strength test can be used as this will both assure the material validity and functionality of the solver. For live testing the target plate material can easily be cut in to the correct shape for a Quasistatic Test on a Instron 8862 or similar machine. In addition, it is the primary material under consideration for this test.

The geometry below was drawn in SolidWorks and imported to Ansys as a STEP file.

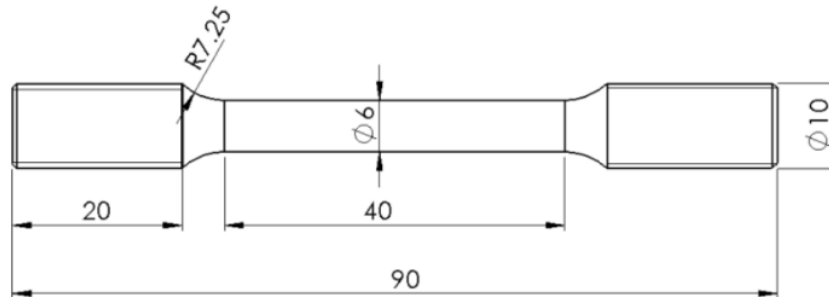


Figure 8 - Geometry and dimensions (mm) of smooth specimen used in tensile test. [6]

The best way to use this test would be to perform a real-world test and compare the results but due to limitations of this exercise this is not possible. Instead, the simulation will be used to generate a stress strain graph that can be compared to the stress strain graph from other real-world testing.

7.1. Material Validation

From this geometry the plate material being used AL5083H116 can be tested for accuracy under the same conditions as the bullet impact scenario. Running the simulation in Ansys explicit dynamics will also provide validation of the chosen simulation model.

The model has been meshed with a Tetrahedral mesh with a density of 0.0001m around the centre section the bar and 0.003 around the thicker ends as show below.

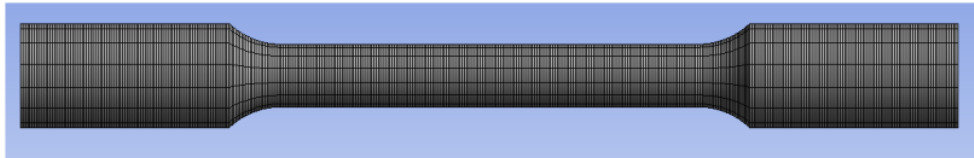


Figure 9 - Test Bar Mesh

A Tetrahedral mesh of this resolution was chosen as a suitable balance between simulation time (15 Minutes) and geometry accuracy. In addition, a more basic mesh failed to provide sufficient accuracy as detailed in Figure 10.

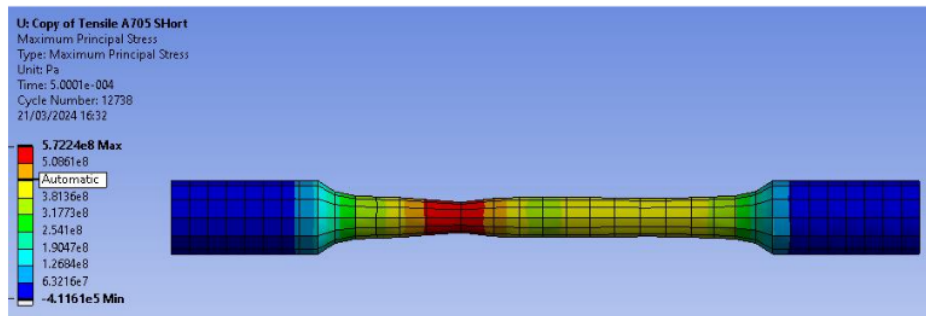


Figure 10 - Basic Mesh Results

In the setup of this simulation all 3 external faces were selected respectively for the fixed and displacement conditions of the bar. This best simulates real world testing where the ends would be placed in clamping collets. It has been assumed that the clamping collets provide no pressure to the bar and the centre test section is sufficiently removed from the collets to prevent this having an impact.

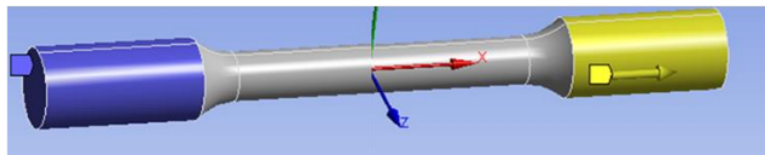


Figure 11 – Fixed and Moving ends respectively.

The visual results of the final test are shown below these indicate the point of slight 'necking' where the material shrinks at a point that is stressed beyond Ultimate Tensile Strength (UTS).

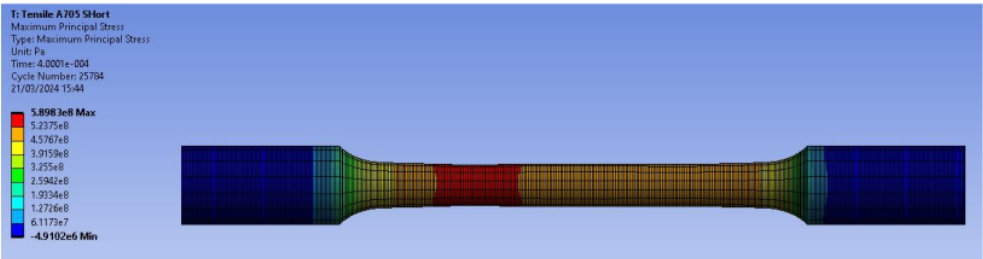


Figure 12 - Stress Concentration

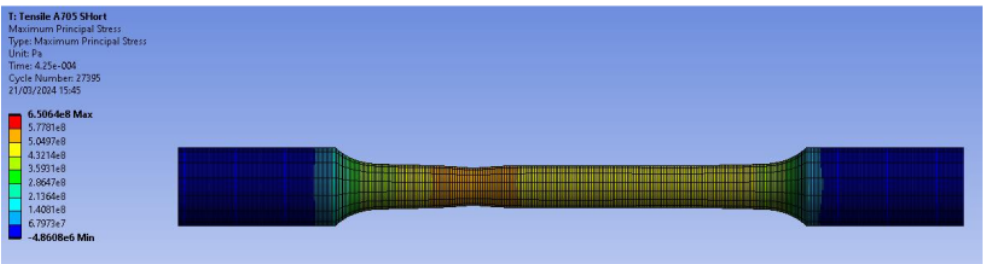


Figure 13 – Necking.

From the Ansys results, data points can be extracted, and a stress strain graph produced as shown in Figure 14. The below graph indicates a UTS of around 544 MPa.

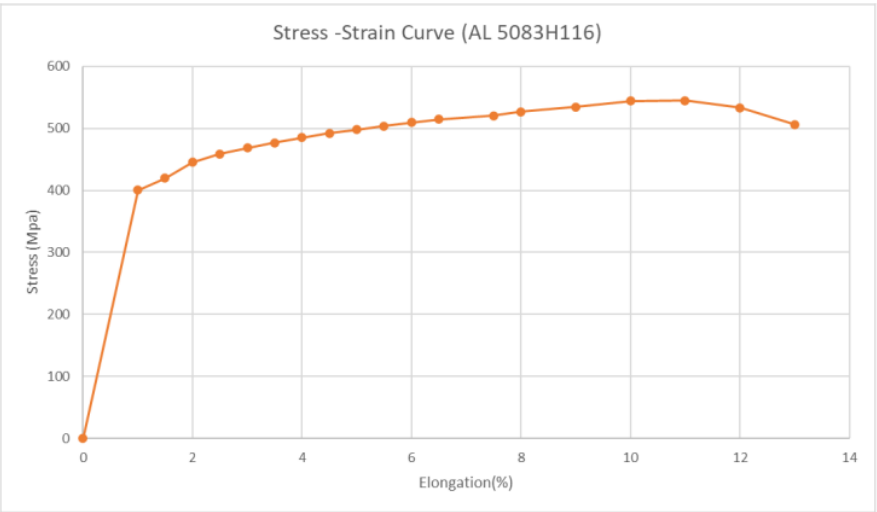


Figure 14 – AL 5083H116 Stress Strain Curve

These results can then be compared to previous experimental data below [7] which supports the simulated data and accuracy of the validation model. The UTS's indicated below put their sample of the same grade aluminium under similar conditions within the same range as the simulated sample performed here. This provides indication the material and solver type chosen will be suitable.

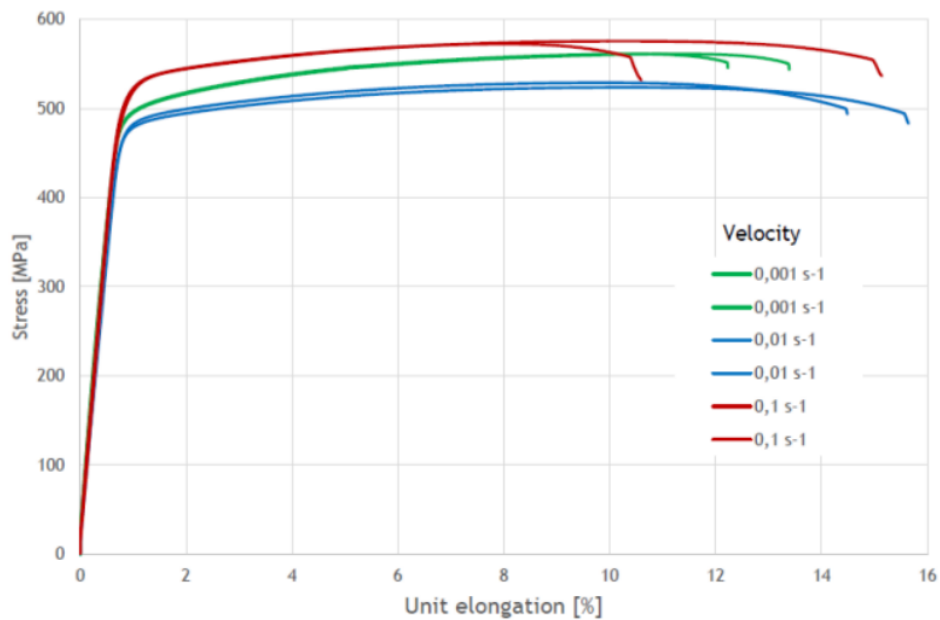


Figure 15 - AL Stress Strain Curve [7]

8. LS dyna vs Explicit Dynamics

There are two main solvers in ANSYS that can be used for this ballistic scenario LS dyna and Explicit Dynamics. Whilst LS Dyna is generally considered a more appropriate solver for a ballistic problem as it completes more complicated calculations at a finer detail, it can prove more complex than necessary for such a simple ballistic scenario.

Explicit dynamics is more suited to simple nonlinear mechanical problems. This is often the case with any CAE or simulation as trade-offs must be made to best suit the needs of the requirements without costing excessively and taking too long. In the case of this scenario explicit dynamic was chosen as the simpler solver allowed for faster iterations of the simulation within the compressed time available, list of simulation variations can be found at Annex A. This decision is a key assumption as using explicit dynamics could produce a greatly different result particularly if the ballistic scenario being considered is too quick.

9. Setup

In the final simulation the explicit dynamics solver was used with geometry, mesh and materials as detailed in previous sections. A velocity of 300 mps was applied to the bullet core and the edge of the target plate was set as a fixed support. The simulation time was set to $1 \times 10^{-4}s$ to allow for sufficient time for the core to impact the plate and penetrate to maximum depth.

10. Results

10.1. Ansys Results

The full results of the simulation can be found at Annex B.

As shown in the full results the bullet impacts the plate and the plate deforms as the bullet penetrates. A stress wave can be observed initiating from the impact point progressing through the target plate and rebounding of the other side. This is classic of a ballistic impact where perforation is not achieved. In the final steps of the simulation the target plate bulges to the rear and the bullet core slightly rebounds as the target plate resists penetration.

The edges of the impact hole show slight mushrooming and deformation as well as indication of cracks starting. Again, this is all classic examples of this type of ballistic impact. The compressive stresses observed throughout this impact reach a maximum of $1.97 \times 10^9 \text{ Pa}$ but from the results it can be observed that a more realistic maximum is around $1.5 \times 10^9 \text{ Pa}$.

10.2. Abaqus Results [1]

From, previous work a very similar scenario of a 7.62mm x 39mm steel core bullet impacting an aluminium plate has been simulated using Abaqus. The materials used in this simulation were almost identical to that of the Ansys simulation allowing for close comparison of broad figures. The full results can be found in Annex C: Abaqus Model 2 Results. These results show the bullet impacting the plate and penetrating but not perforating the aluminium target plate. Initially a tension wave can be observed in front of the projectile as it impacts. As the projectile penetrates further, energy transfer is observed in Figure 29 where significant compression around the projectile is visible. The tension wave can also be seen impacting the back side of the plate causing stretching and deformation. After this last step the round stops and starts bouncing back out leaving the crater in its place. The round does not penetrate the target. In this simulation the maximum compressive stresses observed reach $1.54 \times 10^9 \text{ Pa}$ whilst the tension stresses overcome the material properties in some areas reaching $3.85 \times 10^9 \text{ Pa}$ which is sufficient to destroy and displace the target material.

10.3. Comparison

The results achieved from both the Abaqus and Ansys simulations show a close correlation between each other. Both projectiles penetrated the target plate whilst maintaining their shape and structure. The target plates were the same thickness of 10mm and where both penetrated to a maximum depth of ~7mm. In each case the target plate fractured, displaced, dented and deformed around the projectile bulging towards the rear and becoming deformed as the yield strength was breached. The calculated maximum stresses observed were within a reasonable range of each other indicating a successful simulation on both sides.

11. Limitations and Assumptions

As with any experiment, research or testing assumptions and limitations have to be made and put in place as there is simply no situation where all possible variations, conditions and factors can be considered. At some point the cost will outweigh the benefit.

When considering a simple scenario such as this there are some basic assumptions that get made to further reduce the complexity. For example, in all these simulations it has been assumed that Gravity, Air resistance, and the copper jacket of the projectile have no impact on the final results. In reality, these will have a slight effect. Gravity will pull the projectile of centre; air resistance will slightly slow the projectile and the jacket will impart additional energy supporting in the initial impact. Furthermore, the geometry of the bullet was simplified, the lead cap on the steel core was incorporated into the core itself to simplify the design. Whilst this is inaccurate for the exact example being replicated it is a structure seen in other projectiles such as 7.62mm x 54mm. lastly the ogive of the projectile was estimated from drawings to achieve the final geometry. These slight changes in geometry are assumed to have minimal impact as the material properties and overall size of the steel core are more likely to have a greater impact on penetration depth.

These assumptions were chosen to be eliminated or simplified to streamline the scenario in both simulation time and creation time as every additional variable takes time to input correctly, test that it has been done correctly and adds to the final simulation time.

The only true limitation of using CAE is what the user puts into it as most professional CAE packages can receive large amounts of data and inputs. As with any computer task there is an element of compute power capability to overcome large amounts of data but with the advent of rentable cloud computing space this can largely be overcome through cost. As previously stated with complexity comes compute time, regardless of power.

12. Conclusion

In conclusion the models developed clearly demonstrate a ballistic problem and begin to explore the efficacy of aluminium plate against a likely threat of a 7.62mm x 39mm for the given scenario. In both simulations the results show how the bullet impacts the plate and how the stress within the plate and projectile develops. These simulations also show the limitations of computational modelling as decisions and assumptions must be made to allow for efficiency and workflow. This demonstrates the importance of physical trials alongside the simulations. [1] Due to time limitations the models were not as fully developed as they could have been, material properties could have been further developed and tested. Additional parts of the projectile could have also been included to best represent the scenario. Through the comparison of the two simulations, we are able to gain an understanding of the appropriate solver that can be applied to this simulation. Whilst the Ansys programme allows for a quickly changing simulation with some standard variables and some testing variables, as seen in Annex A, the Abaqus simulation provides slightly more accurate results in a shorter time. Whilst the Ansys toolset is easier to use as a novice the benefits of Abaqus's capability outweigh the ease of use. Overall for this type of simulation Abaqus would appear to be the better choice for a ballistic impact.

13. References

- [1] C. Hadley, *EXPLORING A BALLISTIC PROBLEM WITH COMPUTATIONAL MODELING*, ACDB4010_OMEDeg, UWTSD, SWANSEA: Unpublished Assignment, 2023.
- [2] J. K. Holmen and T. Borvik, "Ballistic impact of layered and case-hardened steel plates," *International Journal of Impact Engineering*, no. 1, 2017.
- [3] R. a. J. R. a. A. D. a. o. Ribeiro, "Modeling and Characterization of Aisi S-7 Steel Using the Johnson Cook," *IOP Conference Series: Materials Science and Engineering*, vol. 1013, p. 012007, 2021.
- [4] L. A. Wuertemberger, "EVALUATION AND CORRELATION OF DYNAMIC FLOW, FAILURE, AND MICROSTRUCTURAL PROPERTIES OF HEAT-TREATED 4130 STEEL," AIR FORCE INSTITUTE OF TECHNOLOGY, Wright-Patterson Air Force Base, Ohio, 2016.
- [5] "Investigation on high-velocity impact performance of multi-layered alumina ceramic armours with polymeric interlayers," *Journal of Composite Materials*, vol. 50, 2015.
- [6] R. T. DE Tria, "Dynamic Characterization and Constitutive Modelling of ARMSTAL 500 Steel," Military University of Technology, Warsaw, Poland, 2014.
- [7] E. R. R. S. J. S. a. B. K. Paweł Balon*, "Implementation of high speed machining in," *Open Engineering*, vol. 8, no. 1, pp. 162-169, 2018.
- [8] D. E. T. a. R. Trebinski, "ON THE INFLUENCE OF FRACTURE CRITERION ON PERFORATION OF HIGH-STRENGTH STEEL PLATES SUBJECTED TO ARMOUR PIERCING PROJECTILE," *Archive of Mechanical Engineering*, vol. LXII, no. 2, pp. 157 - 179, 2015.
- [9] E. Flores-Johnson, L. Shen, I. Guimatsia and G. Nguyen, "Numerical investigation of the impact behaviour of bioinspired nacre-like aluminium composite plates," *Composites Science and Technology*, vol. 96, no. 1, pp. 13-22, 2014/05/01.
- [10] MatWeb, "Aluminium 7075-T6;7075-T651," MatWeb, [Online]. Available:
] <https://www.matweb.com/search/DataSheet.aspx?MatGUID=4f19a42be94546b686bbf43f79c51b7d>. [Accessed 29 04 2023].

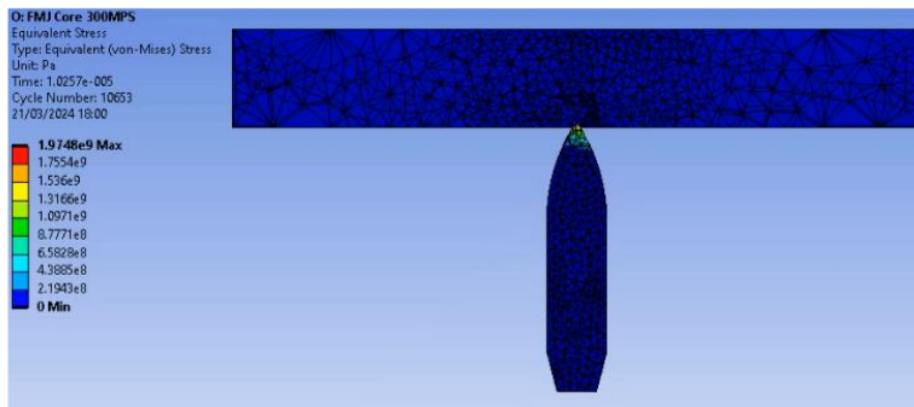
15. Annex B**16. 300mps Ansys Explicit Dynamics Side Cross Section view**

Figure 17 – Step 1

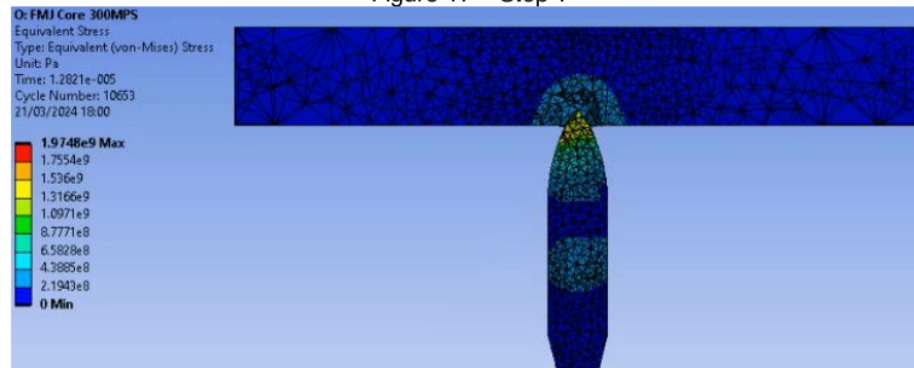


Figure 18 – Step 2

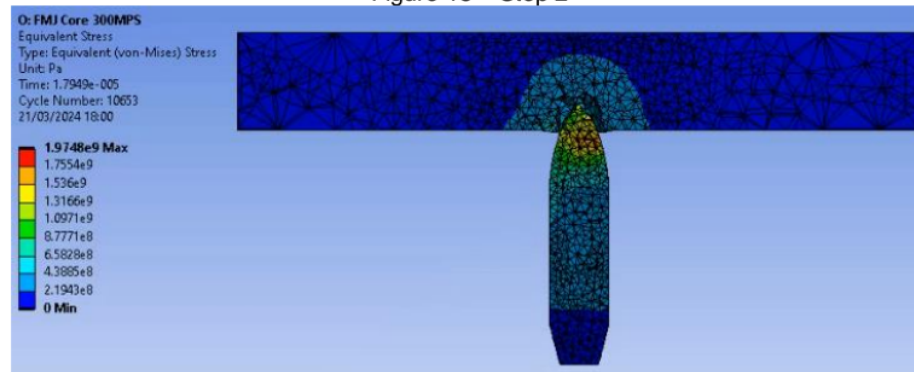


Figure 19 – Step 3

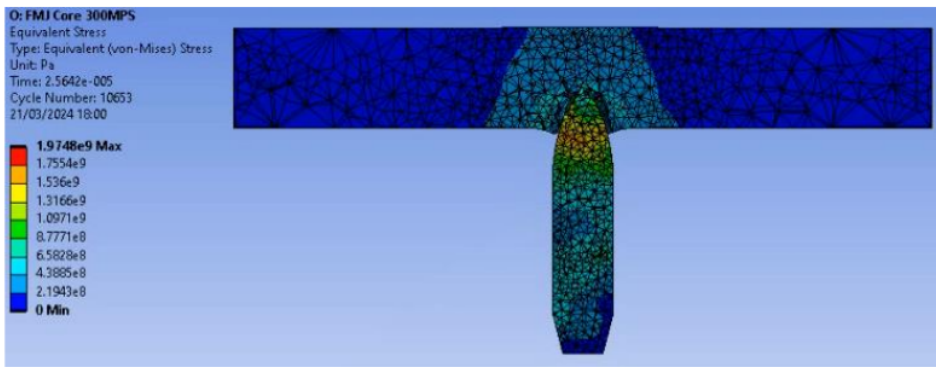


Figure 20 – Step 4

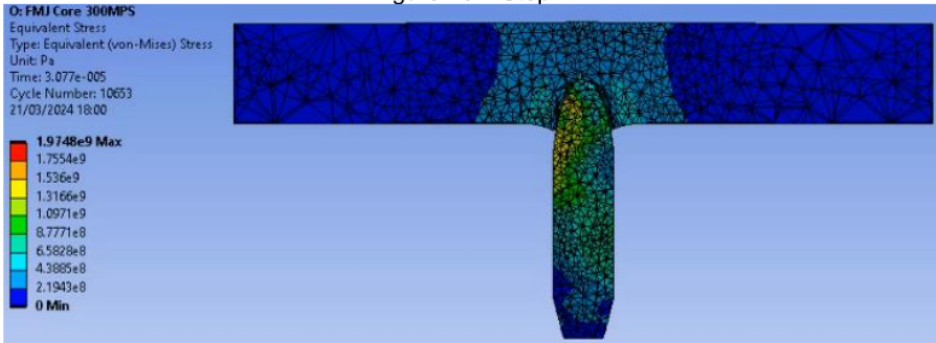


Figure 21 – Step 5

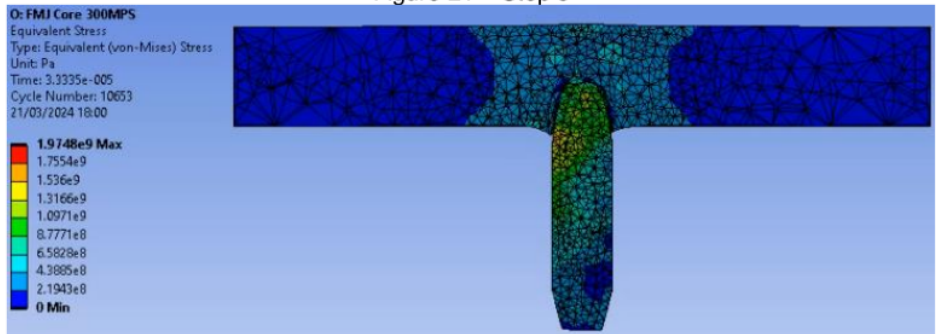


Figure 22 – Step 6

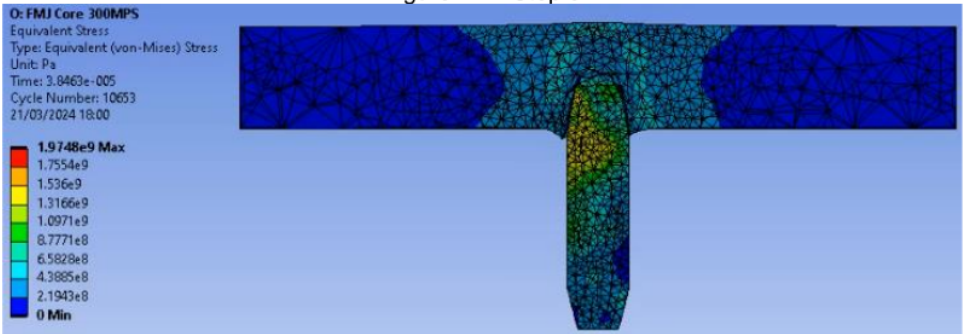


Figure 23 – Step 7

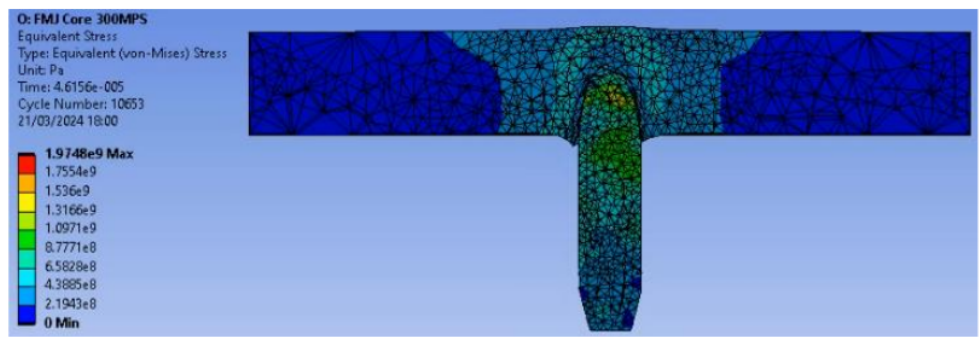


Figure 24 – Step 8

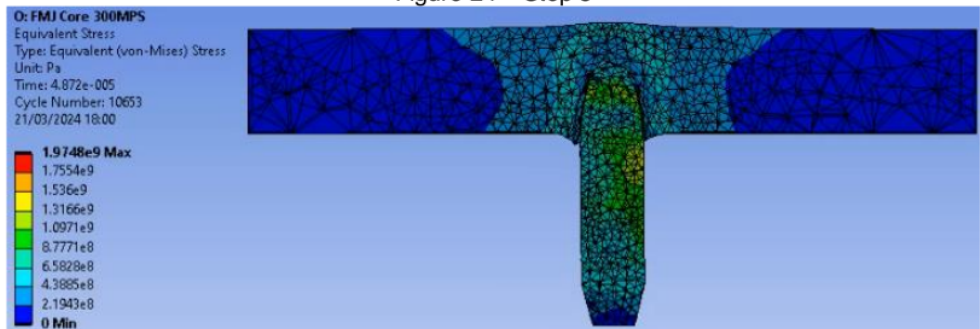


Figure 25 – Step 9

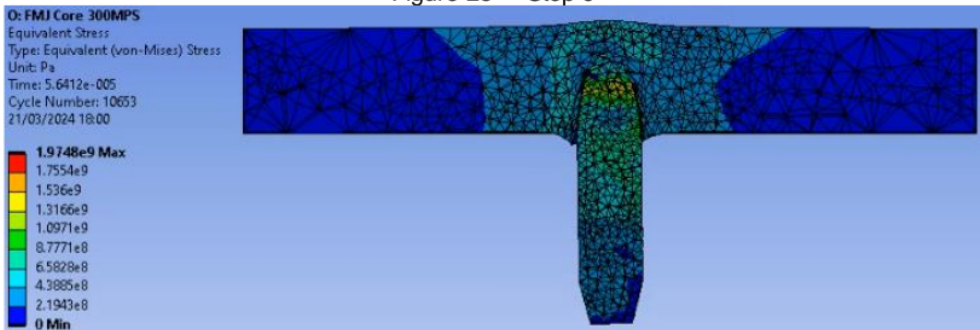


Figure 26 – Step 10

17. Annex C: Abaqus Model 2 Results [1]

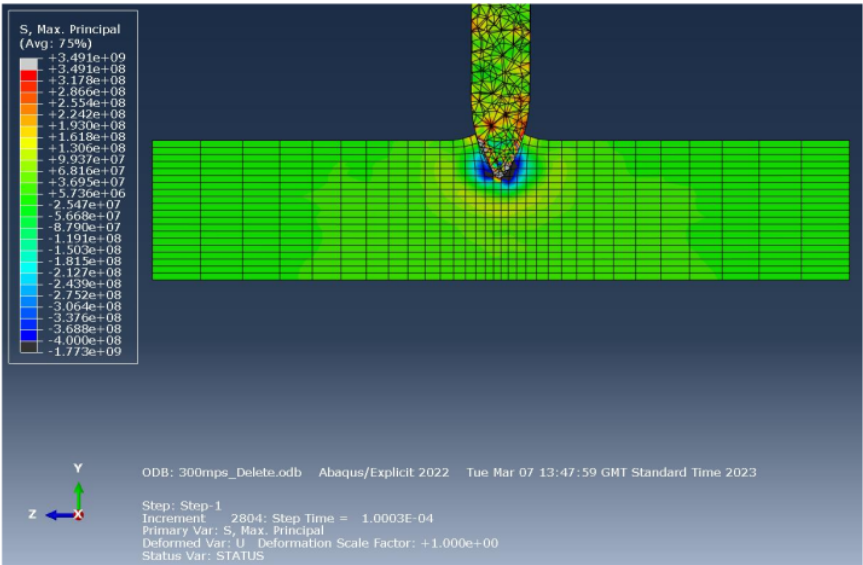


Figure 27 – Model 2, 300mps, Step 1

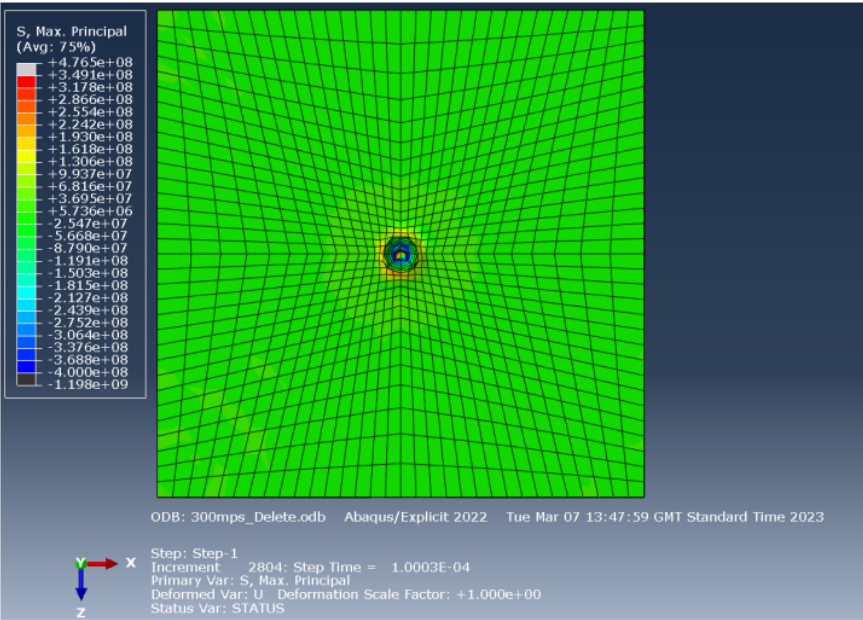


Figure 28 – Model 2, 300mps, Step 1, Top

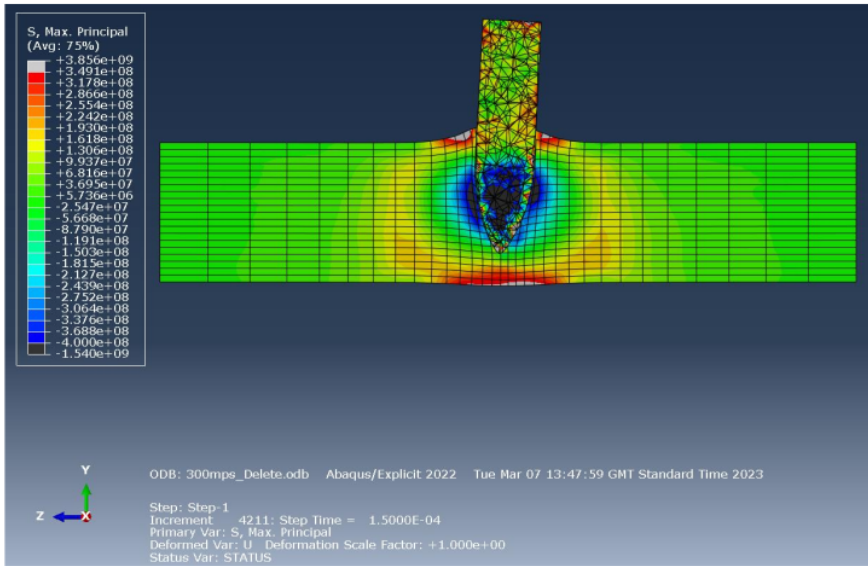


Figure 29 – Model 2, 300mps, Step 2

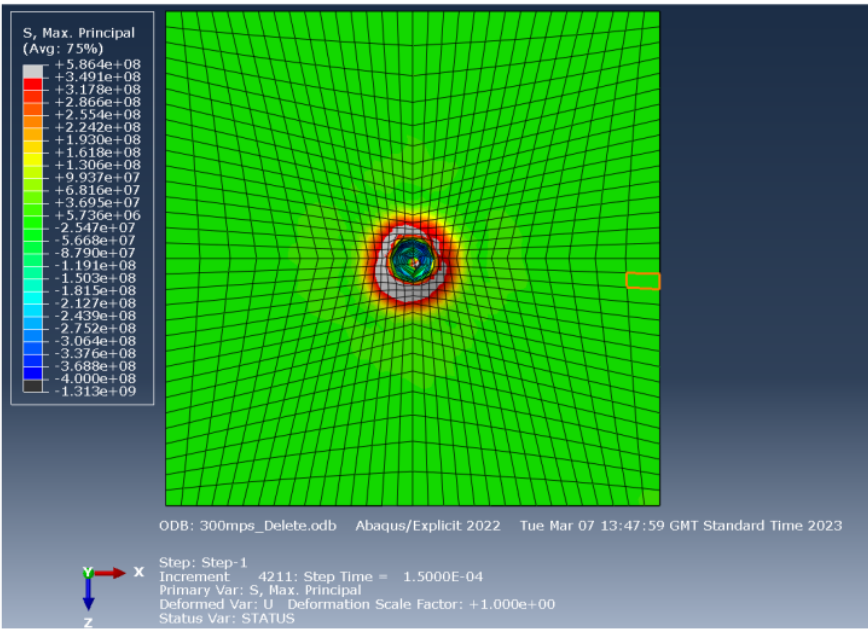


Figure 30– Model 2, 300mps, Step 2, Top

FINAL GRADE

66/100

GENERAL COMMENTS

A reasonable context is given at the start of the report that gives rational to the study. You noted some of the assumptions and simplifications that would be made in the simulations, it would have been nice to have had some more extensive detail and supporting information to better justify these decisions.

Your geometry creation was sensible based on the dimensions shown from the literature source, and the target plate sizing looked reasonable. Where you stated the target size was based on practical experience, this in itself is fine, although a bit more explanation of that would have been useful to highlight why it is important and what the implications are of having target sizes that are smaller/larger.

Meshing decisions could have had more supporting information to go with them. For example, a simplified test case to show how different mesh types influence the output would have been good to demonstrate this. You do go on to do some of this in the validation section though, which a good start.

The validation approach taken was a very good one, creating a scenario that replicates understood testing methods that are intended to put a material well beyond its linear elastic range. The model has responded well to that and you have used the output data nicely to correlate against the material properties and it seems to have worked to a good level of accuracy. It would have been nice to have quoted and compared the specific values, or had your data overlaid onto the material data you obtained, to make the comparison more obvious, but what you had here was still very good.

Some interesting comparison of Dyna and ED, you could have gone into further depth here, but the approach of comparing the options was good.

In your results, it would have been preferable to have included some screenshots of the generated data to make this section easier to understand and interpret for the reader. They were in the appendix, but be warned in future that an appendix and information contained within it does not have to be looked at, it is best to keep key information in the main report where appropriate.

The generated results looked promising, and the 2 software's did generate similar results which was also encouraging. It would have also been good to have then compared your results against similar published studies, to see if the trends were matching to other peoples data.

You have had a good attempt at discussing the limitations of the simulations and hypothesizing as to how some of these may affect the outcome. This is an area that could have been explored further, but was good to see the approach taken.

A generally well presented document, although the reliance on appendices is something you might want to adjust in future to make sure that critical information is found in the most relevant areas.

A reasonable number of sources have been employed and used well in the report. There was scope to have had more comparative analysis to literature, but generally decent.

PAGE 2

PAGE 3

PAGE 4

PAGE 5

PAGE 6

PAGE 7

PAGE 8

PAGE 9

PAGE 10

PAGE 11

PAGE 12

PAGE 13

PAGE 14

PAGE 15

PAGE 16

PAGE 17

PAGE 18

PAGE 19

PAGE 20