

2113123 - CAE

by Stephen Mcangus

Submission date: 01-May-2023 09:27PM (UTC+0100)

Submission ID: 204169402

File name: 33008_Stephen_Mcangus_2113123_-_CAE_497865_2016835969.docx (267.25K)

Word count: 4013

Character count: 22165



Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

ACDB5007 – L5 – Computer Aided Engineering

Student: Stephen McAngus (2113123)

Abstract:

Computer aided engineering and finite element analysis are tools used by industry to exploit computer processing to generate simulations on virtual structures that represent real structures, or engineering systems. The data captured by this study shows that both computer aided engineering and finite element analysis remain a viable tool in forecasting the effect of a projectile hitting a target. As such, this tool can be used to predict the behaviours of a scenario prior to trailing, or prototyping concepts. However, neither tool should be used by itself to fully validate a concept, and in most circumstances where there is an aspect of danger to life, physical trials should also be implemented to validate the results obtained in a software package.

Contents

Introduction:..... 3

Method and results:..... 5

Figure 1: displays the dimensions of a 5.56 NATO round. This round is the 85.5-grain munition produced by Berger Bullets, dimensions of this round are imperial, and were converted to metric prior to input into SolidWorks [reference]..... 5

Figure 2: Displays the Shear Elastic Strain on both an 85.5 Grain 5.56 Projectile formed of Tungsten, and a Target composed of Aluminium. When the projectile is travelling at 955.5m^{-2} 6

Figure 3: Top image displays the total deformation on both the Projectile (Tungsten) and Target (aluminium), with relation to the angular motion of force after the effect. Bottom image displays the total deformation of the Projectile (Tungsten) and Target (Aluminium), bullet velocity was set to 955m^{-2} matching the muzzle velocity of an 85.5 Grain 5.56 projectile for both runs. Red particulate dots represent element deletion. 7

Figure 4: Displays the resultant velocity of the Tungsten projectile after traveling at 849.9m^{-2} . This also displays the directional velocity of both projectile and Aluminium target post impact. During the simulation the intensity of the velocity of the target changed to a peak of 106.16m^{-2} 8

Figure 5: Displays a simple static structural cantilever validation study for the material properties of the reference aluminium material used in the ballistic target, under 50N of force. 8

Discussion 9

References 13

Introduction:

Computer aided engineering is as implied by the name, the exploitation of computer processing power to produce a means of generating or analysing a complex engineering task. Computer aided engineering is a relatively modern concept introduced in the 1970's by the structural dynamics research corporation [1]. Since inception, computer aided engineering has been utilised by several sectors within the engineering industry, such as structural engineering, automotive, and defence [2] [3] [4] [5] [6]. Computer aided engineering is responsible for the streamlining of engineering processes, where one competent person and a computer can now complete projects that would once take multiple teams of personnel. Not only has computer aided engineering had an economical benefit to industry, but it has also led to a significant reduction in time to process significantly challenging engineering tasks [4] [5].

Most computer aided engineering programmes work by exploiting different sets of data against one another. Nowadays computer aided engineering specifically exploits the geometric properties of a three-dimensional design that is created through computer aided design tools. The market is flooded with over 100 different computer aided design tools, however for the purposes of this work, Solidworks was used, as it has been proven to be an industry standard [7] [8] [9]. Computer aided design tools typically work on the basis of producing a two dimensional sketch of an object, and then using tools to form bodies or features. Over many years of development, computer aided design software packages have become increasingly complex for the betterment of industry. Some of the first computer aided design packages were only used for the three dimensional design process. However, software such as Solidworks can run rudimentary computer aided engineering systems, such as analysis of stress and strain, but more importantly interactions of whole systems.

Using computer aided design in line with computer aided engineering assists an engineer with understanding the full picture of a concept. For example, an engine can be designed in computer aided design, and using some of the features of SolidWorks the interactions of features and components can be isolated. These interactions can then be analysed through very complicated mathematical tools. In this instance, analysing the stress and strain of a crank shaft could be conducted through Finite element analysis. Finite

element analysis tools such as ANSYS provide a means of analysing the geometric and material properties of a structure, against external factors, such as heat, friction, load, and impact.

Finite element analysis works on the principle of discretizing a complex geometry, into smaller, but simpler geometries and then totalling the result from the breakdown process. This process is known as meshing, meshing is when an object is broken down into smaller sections, to a simpler shapes, by default these tend consist of a cuboidal geometry. however, there are instances such as when there are more rounded features that other meshing geometries may be more appropriate, such as tetrahedral or prisms [10]. Furthermore, a mesh is also assigned material data; this material data encapsulates the physical properties of a material, such as ductility, elasticity, heat capacitance, density and fracture hardness. After assigning material properties to the meshed component and engineer can establish what load can be applied to the system. The finite element analysis package, in this instance ANSYS will run simulation of the scenario specified by the engineer and mathematical formulae will solve for what happens to the individual nodes of the mesh. The combination of the resultants from the individual nodes produce an overall view of what happens to the entire component or system.

There are a few approaches of finite element analysis, static, explicit and dynamic. Static studies are mathematical equations that are reliant on the equilibria of a system, i.e. displacements under load, typically this type of mathematics is achieved through matrices [11] [12]. An explicit study is the mathematical method where force is integrated over steps of time, often this approach studies small segments of time to view a linear scale of events [13]. Whereas a dynamic approach is a time dependant study of a variety of effects upon a system, this is achieved through the use of differential equations to show nonlinear trends [14].

The aim of this study is to evaluate whether or not finite element analysis is a suitable tool for engineering purposes, and to establish if finite element analysis can be effectively deployed in a ballistics scenario. With the objectives of designing a system to study through the use of computer aided design, to then be exploited through a dynamic finite element analysis study within ANSYS. And to validate the results that were captured from the study.

Method and results:

Following publically available designs a projectile representing a 5.56 NATO Standard munition was designed in SolidWorks. Initially this design was to be jacketed and to have a penetrative core, however due to processing limitations this concept was dropped from the design, and a full body projectile was adopted. A simplistic cuboidal target was also formed, these two components were then assembled in SolidWorks and were then exported to a file format suitable for ANSYS.

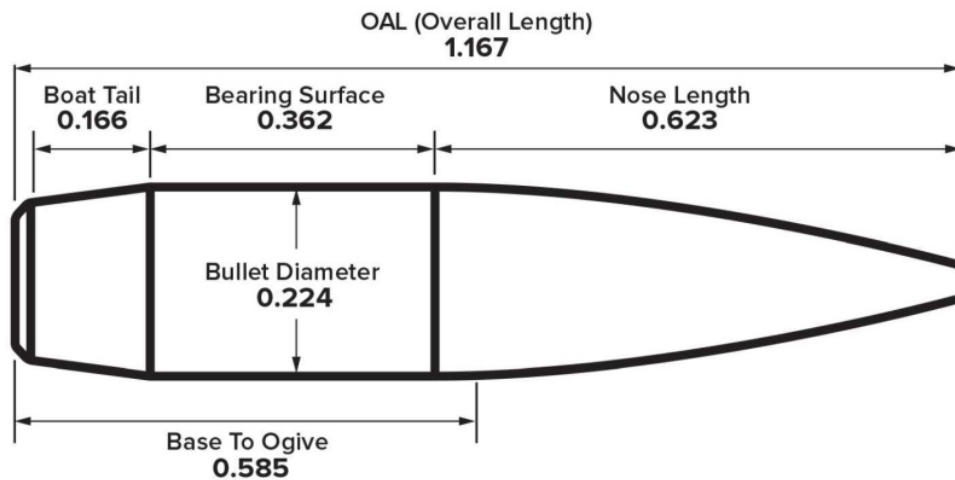


Figure 1: displays the dimensions of a 5.56 NATO round. This round is the 85.5-grain munition produced by Berger Bullets, dimensions of this round are imperial, and were converted to metric prior to input into SolidWorks.

Once the model had been imported into ANSYS, an automated mesh was applied to both the projectile and the target features. Both of these meshes were then refined to a finer quality so that some tangible results could be obtained. Configuring ANSYS to an Explicit Dynamic study allowed for an assessment of a projectile moving and hitting a target at a referenced speed of 955m^{-2} muzzle velocity. This velocity was applied to the projectile traveling along the X axis, in the direction of the target. Using the ANSYS Material catalogue, a Tungsten reference property profile was assigned to the projectile. An aluminium reference property profile was assigned to the target, refer to Table 1, annex. Once the profile was established, three resultants were selected, Shear Elastic Strain, Total Deformation and Velocity.

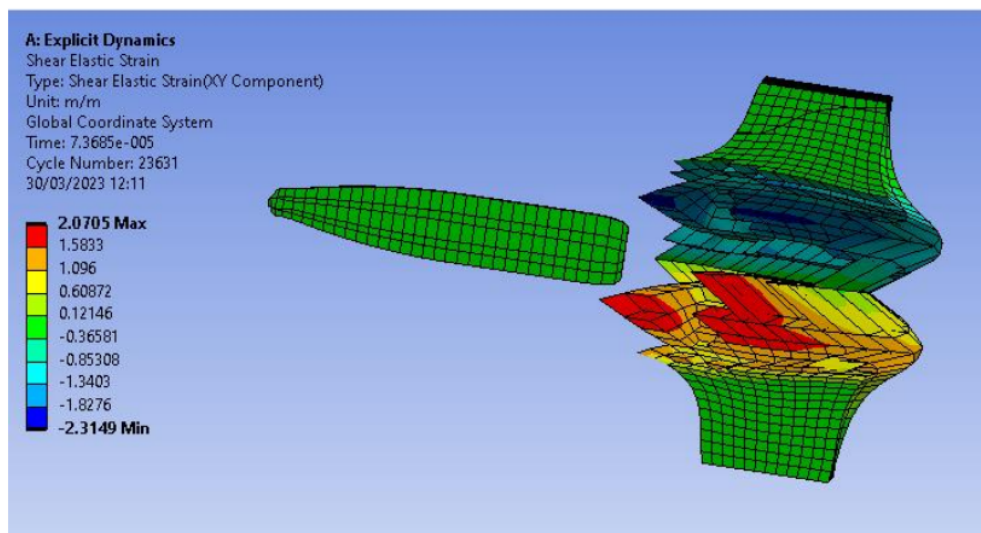


Figure 2: Displays the Shear Elastic Strain on both an 85.5 Grain 5.56 Projectile formed of Tungsten, and a Target composed of Aluminium. When the projectile is travelling at 955.5m^{-2} .

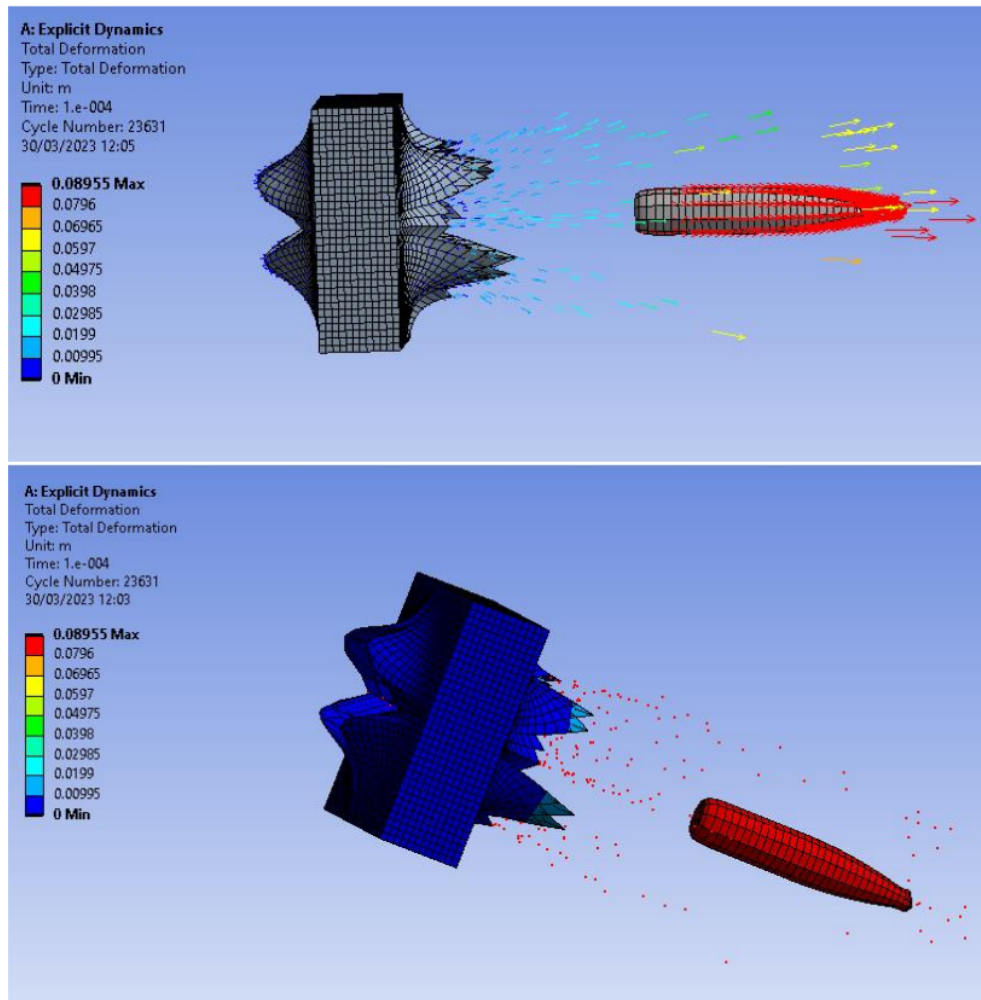


Figure 3: Top image displays the total deformation on both the Projectile (Tungsten) and Target (aluminium), with relation to the angular motion of force after the effect. Bottom image displays the total deformation of the Projectile (Tungsten) and Target (Aluminium), bullet velocity was set to 955m^2 matching the muzzle velocity of an 85.5 Grain 5.56 projectile for both runs. Red particulate dots represent element deletion.

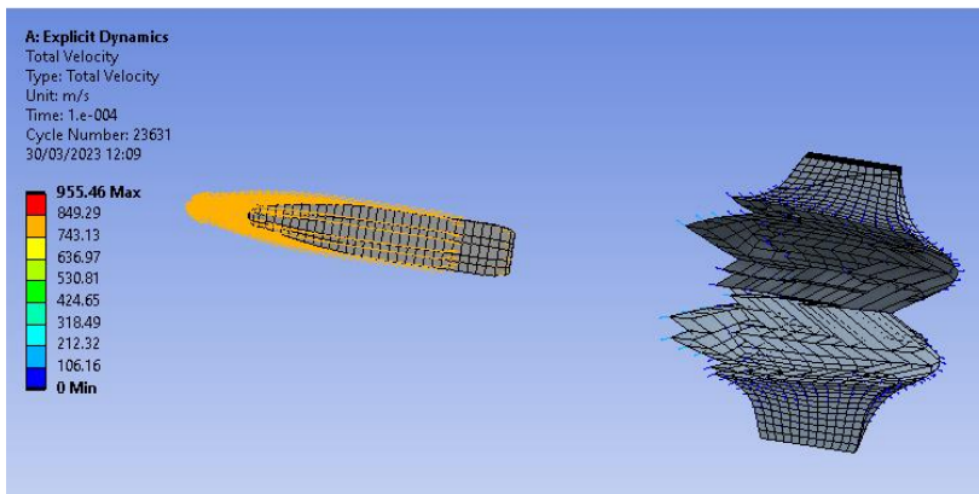


Figure 4: Displays the resultant velocity of the Tungsten projectile after traveling at 849.9m^{-2} . This also displays the directional velocity of both projectile and Aluminium target post impact. During the simulation the intensity of the velocity of the target changed to a peak of 106.16m^{-2} .

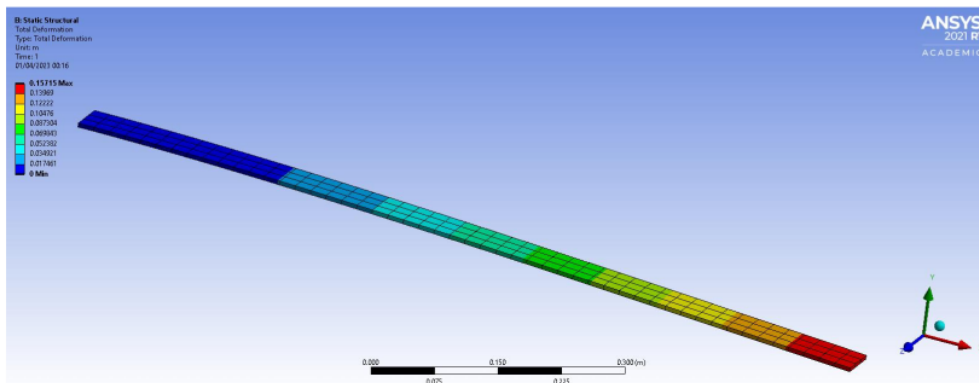


Figure 5: Displays a simple static structural cantilever validation study for the material properties of the reference aluminium material used in the ballistic target, under 50N of force.

Discussion

Total deformation showed that there was both deformation on both the projectile and the target. The target was fully penetrated, and the full deformation as displayed in the results, this was expected as aluminium is a very ductile material. Whereas the projectile formed of tungsten sustained very little damage, this matches the performance properties of tungsten. However, to that end, the colour scale of the projectile suggests that this projectile was undergoing large levels of stress which can be expected given the force it was experiencing. However, this may also be a meshing issue, due to meshes not being suited to fine, or acute angular features. The interpretation of the deformation profile is also supported by the elastic properties measured in the simulation. What can be seen is that the projectile throughout the simulation did not experience any elastic strain, whereas the malleable aluminium target stretched until failure. Measurements of velocity showed that the target had a very negligible effect on the projectile, reducing final speed from initial speed by approximately 200m^{-1} . It was however impossible to extrapolate from this data set as the effect impact and friction had upon the transition through the target.

The experimental design for the most part provided a very basic understanding to the use of finite element analysis for ballistics. flaws in the design began from the computer aided design stage, where designing a jacketed munition proved to be near impossible with the very acute angles, and sharp radials involved, due to meshing errors, this concept had to be dropped from this project. Experimentation was undertaken to refine the mesh in localised areas, and to introduce a tetrahedral mesh at the tip of the projectile, however the mesh density also caused significant errors in calculation. These errors were experienced as node deletions without imparting any deformation or stress to the target. As such, this model was not fully representative of an armour piercing small arms munition. Furthermore, it was identified that several other factors could have been implemented to this simulation to provide greater detail on this collision. Rotation of the projectile, or spin imposed from rifle is thought to have an effect on penetrative power of a small arms round, as such, future study should include this.

In-part due to the limitations of ANSYS and other finite element analysis software packages some of the results are not fully representative to what can be observed in physical studies. most of these issues can be attributed to meshing errors, the accuracy of the results obtained from an finite element analysis study is highly dependent on how well the mesh has been refined, and or if the mesh type is suitable for the purpose of study [15]. This statement does not reflect that all meshes must be of a finer quality, but proportional. Inherently by refining the mesh sizing, the number of rounding errors increases proportionally to the refinement. This is because there are a significantly greater number of nodes to calculate data for, and as such the software will round results to reduce impact upon processing. As suggested, by increasing the amount of calculations by refining the mesh will also lead to significantly longer processing times, thus there is a more practical reason as to why less refined meshes are used. The advent of Quantum Computing may assist with the negation of rounding error from meshing. On the principles of bits being in more than one state at any one time this gives the ability to calculate an entire model, rather than a simplified geometrical mesh structure.

Furthermore, if meshing is not negated, then quantum computing will provide significantly faster processing time, and as such would make a system more capable of calculating complex mathematics. The geometry that was imported into ANSYS from this study was a relatively complex geometry, with differing radius' which translated to there be a potential for meshing distortion, or convergence errors arising. Meshing distortions commonly arise when meshing a feature with a number of differing radius's [10]. Furthermore, by increasing the complexity of this mesh, the likelihood of convergence errors arising significantly increases [10]. To negate this advanced meshing could be applied to the structure, this tends to be executed by hybridising different mesh types together, to best fill the gaps of the original geometry, i.e. by mixing a tetrahedral and cuboidal mesh together to refine a curved surface in greater detail, and also makes the data more representative.

It is important to note that most of the failings with computer aided engineering and finite element analysis are more intertwined with the human interference rather than the software. Knowledge, and experience severely limit the ability to deploy these systems. To properly utilise, and effectively deploy ANSYS or similar finite element analysis tools, a user

requires very in-depth training and knowledge. This knowledge and expertise also ties into understanding the assumptions made within finite element analysis. Assumptions can be made on most interfaces within the software, such as material properties, assumptions may lead to a trend of thought that a material may do one thing in one scenario, but when physically testing this concept prove drastically different results. to that end, a means of negating the effect of improper training and inexperience would be to utilise artificial intelligence, combining this with quantum computing would lead to a system being able to make judgements on how to best execute an finite element analysis study, and to pre-validate the parameters of a study [16].

Validation for the results obtained on this project were outside of the five percentile tolerance. Several factors attribute towards why there is such a variance, the difference in study type is the leading cause for this differentiation. The validation study chosen was a static structural study, whereas all other results obtained originated from Explicit Dynamic studies. The material data required for either of these tests can be drastically different, as such the results are incomparable, and no conclusions can be extrapolated. To that end, analysing the convergence solutions would be used in future work to validate the results obtained [17] [18]. Other contributing factors for difference in between validation and other results can be attributed towards mesh sizing. The mesh between the two studies were not proportional to one another, and as such there are more rounding errors obtained from the Explicit Dynamic study, than in the validation study.

This study has provided results that display there is plausibility for finite element analysis to be utilised in a manner to prove a concept. However aforementioned there are some indicators that this exact study does not fully represent as to what could be observed with a physical study. Using a Multiple Body Dynamics study could have provided more stringent results from the data captured in this study. Multiple body dynamics is a dynamic study tool kit that can accurately plot the relationship between a mechanical systems in a dynamic setting. In this instance, a multiple body dynamics study could have provided greater insight into the transient properties of the target and projectile as the two objects collide. However, multiple body dynamics and finite element analysis are both very assumptive tools, for example neither study would encapsulate microdata, such as friction induced upon a projectile when in flight, environmental temperature, or energetic decay.

The impact of these factors for the purposes of engineering are however minor, and as such could be considered insignificant in the total orders of magnitude.

Finite element analysis tools tend to be a very taxing software's for a computer to run, using large amounts of system memory, and processing power to complete a complicated study. As a result a user may have to simplify a model significantly to reduce the processing time involved with these studies. Furthermore, this may have a knock on effect with the accuracy of the results obtained. To that end, there are instances where a model can be segmented and each component to be analysed independently. Although a great idea of paper, and providing strong results for each of these components under certain stresses, this does not translate to how well a whole system may operate. To mitigate the use of short cut approaches, it is common for many establishments to run these simulations on data centres, where processing power and system memory become more negligible. Due to current limitations with processing finite element analysis and other engineering simulation tools there is a limitation as to what can be simulated [19] [20]. To mitigate this parallel processing would aid in the number crunching involved in complex simulations. Simply it breaks down a complex simulation across multiple processors, or multiple systems, yielding a significantly faster processing time, and greater ability to analyse significantly complex structures [21] [22].

Computer aided engineering and finite element analysis both serve as fantastic tools to the field of engineering, and allow for the forecasting and predictive testing prior to the establishment of prototyping, or implementation of a trial. However, it is drawn back by the fact that these study types can only calculate from the data the system understands. As of recent, new mathematical models have been implemented to analyse the statistical significance of results obtained from finite element analysis, to mitigate the uncertainty that can generate in a simulation [23] [24]. Predominantly there are two forms of uncertainty analysis, Aleatory and Epistemic, Aleatory works on the principle of quantifying variability and factoring this into the output solution of a simulation, somewhat a confidence level in results [25] [26] [27]. Whereas Epistemic focuses more on prediction of lacking information. As such the finite element analysis software can make informed predictions on the math, and reflect this in the output resultant [26] [28].

In conclusion, computer aided engineering and finite element analysis both provide industry a means of proving concept prior to developing prototype, this in turn assists in reducing the total cost of the research and development of products. Whilst it remains a very strong means of understanding the effects of different forces against a model and the associated material properties, it is limited predominantly by the information the software is given, meshing error, and convergence errors. Finite element analysis should not be used to fully prototype and develop a design, and the results from the finite element analysis study should be validated by physical studies, especially if the design in question poses a risk to life, or reputation.

References

- [1] P. Marks, "2007: In remembrance of Dr Jason A Lemon, CAE pinoneer", GFSpeak.com, 2007.
- [2] L. Egab, "Chapter 10 - Computer Aided engineering findings on the physics of tyre/ road noise," in *Automotive Tyre Noise and Vibrations, Analysis Measurement and Simulation*, Butterworth-Heinemann, 2020, pp. 217-243.
- [3] D. Fuchs, "Necessary advances in computer-aided design to leverage additive manufacturing design freedom," *International Journal on interactive design and manufacturing*, vol. 16, pp. 1633-1651, 2022.
- [4] J. Liu, "Computer-Aided Design-Based topology optimization System with Dynamic Feature Shape and Modeling history Evolution," *The American Society of Mechanical Engineers*, vol. 142, no. 7, p. 13, 2019.
- [5] K. Hyuk-Jae, "Computer aided engineering (CAE) simulation for the design optimization of gate system on high pressure die casting (HPDC) process," *Robotics and Computer-Integrated Manufacturing*, vol. 55, pp. 147-153, 2019.
- [6] S. Rezwana, "A review of CAD to CAE integration with a hierarchical data format (HDF)-based solution," *Journal of King Saud University - Engineering Sciences*, vol. 33, no. 4, pp. 248-258, 2021.
- [7] N. Bratovanov, "Robot Modeling, Motion Simulation and off-line Programming based on Solidworks API," in *Third IEEE International Conference on Robotic Computing (IRC)*, Naples, 2019.
- [8] A. Bocevskaja, "An Example of Application Design Using SolidWorks Application Programming Interface," in *Information Systems and Grid Technologies*, Sofia, 2020.

- [9] J. R. Ottway, "An Investigation into the Value and Benefits of the SolidWorks Certification program," in *2019 ASEE Annual Conference & Exposition*, Tampa, 2019.
- [10] Y. Hu, "Fast Tetrahedral meshing in the wild," *ACM Transactions on Graphics*, vol. 39, no. 4, pp. 1-18, 2020.
- [11] S. R. D. Galarreta, "A validated finite element analysis procedure for porous structures," *Materials & Design*, vol. 189, 2020.
- [12] V. Roda-Casanova, "A 2D finite element based approach to predict the temperature field in polymer spur gear transmissions," *Mechanisms and Machine Theory*, vol. 133, pp. 195-210, 2019.
- [13] Z. Yang, "Numerical Modeling of Dynamic Frictional rolling contact with an Explicit finite element method," *Tribology International*, vol. 129, pp. 214-231, 2019.
- [14] J. Wang, "Analysis method of water inrush for tunnels with damaged water resisiting rock mass based on finite elelment method-smooth particle hydrodynamics coupling," *Computers and Geotechnics*, vol. 126, 2020.
- [15] J. Yang, "Reinforcement learning for adaptice mesh refinement," in *26th International Conference on Artificial Intelligence amd Statisitcs*, 2023.
- [16] D. Hackett, "Digitizing gauge fields: Lattice monte carlo results for future quantum computers," *Physical Review*, vol. 99, 2019.
- [17] R. Carmona, "Convergence analysis of machine learning algorithms for numerical solution of mean field control and games: II - the finite horizon case," *Ann. Appl. Prohab*, vol. 32, no. 6, pp. 4065-4105, 2022.
- [18] T. Doan, "Finite-Time analysis of Distrabuted TD(0) with linear function approximation on Multi-agent reinforcement learning," in *Proceedings of the 36th International Conference on Machine Learning*, 2019.
- [19] L. Chen, "Structural Shape optimization of three dimentional acoustic problems with isogeometric boundary element methods," *Computer Methods in applied mechanics and engineering*, vol. 355, pp. 926 - 951, 2019.
- [20] D. C. G, "The Use of finite element methods for design with adhesives," *Measurement Good Practise Guide*, vol. 48, no. 2, 2023.
- [21] Q. Wu, "Parallel Computing in Railway research," *International Journal of Rail Transportation*, vol. 8, no. 2, pp. 111-134, 2018.
- [22] D. Marinkovic, "Survey of Finite element method based Real time simulations," *Department of Structural Analysis, Technical University of Berlin*, vol. 9, no. 14, p. 2775, 2019.
- [23] S. Ghosh, "Uncertainty analysis of microsegregation during laser powder bed fusion," *Modelling and Simulation in Materials Science and Engineering*, vol. 27, no. 3, 2019.

- [24] K. Gao, "Hybrid uncertainty analysis of functionally graded plates via multiple-imprecise-random-field modelling of uncertain material properties," *Computer Methods in Applied Mechanics and Engineering*, vol. 368, pp. 113-116, 2020.
- [25] P. Castaldo, "Aleatory uncertainties with Global resistance safety factors for non linear analyses of slender reinforced concrete columns," *Engineering Structures*, vol. 255, p. 113920, 2022.
- [26] M. Amin, "Efficient seismic reliability analysis of large-scale coupled systems including epistemic and aleatory uncertainties," *Soil Dynamics and Earthquake Engineering*, vol. 116, pp. 761-773, 2019.
- [27] M. Dannert, "Non-linear Finite Element Analysis Under mixed Epistemic and Aleatory Uncertain Random Field Input," in *29th European Safety and Reliability Conference*, Hannover, 2019.
- [28] J. Chen, "A compatible probabilistic framework for quantification of simultaneous uncertainty of basic parameters of structures by synthesizing the change of measure and change of random variabilities," *Structural Safety*, vol. 78, pp. 76-87, 2019.
- [29] Z. Yang, "Numerical Modeling of Dynamic Frioctional rolling contact with an explicit finite element method," *Tribology International*, vol. 129, pp. 214-231, 2019.

FINAL GRADE

68/100

GENERAL COMMENTS

Instructor

Abstract gives a reasonable overview of the work presented.

Contents page seems to have gone a bit wrong! Think you have not got the section headings and captions correct in the document.

There is a good range of citations used in the report, with some good academic quality ones used which was good to see.

The introductory sections of the report give a nice contextual overview of the uses of CAE and the general process as it applies to this context. There were some areas that could have had a bit more discussion and support, but generally good.

The report structure made the information a little hard to follow. A better approach would have been to have clearly split the report into more defined sections. Presenting the validation works in their own section and then analysing them would have made that clearer. Similar with the main study, it would have been better to have presented and analysed the results for each study in their own sub-sections rather than having to scroll up and down the report to refer back to the results images for the different study iterations.

The post-processing shown in the report could have been clearer and more targeted. For example, you mention about the velocity plots of the projectile and how the results were a bit abnormal, but you did not show the plots in the main report

as a graph, only a screenshot of a singular time step.

You discussed the issues with the geometry in your early attempts, would have been good to have shown the failed results from these studies to better illustrate the points you were making and justify the decision to change direction with the geometry being used.

You had some very valid discussion about the impact of meshing on results, with literature support, but it would have been nice to have seen you conduct your own study into this, even if on the validation case. Supporting your statements with data is always the better way of justifying the argument presented.

You have noted some of the significant known limitations of use of CAE and had some good discussion around these areas, although these could have been contextualised to your own studies more.

PAGE 1

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
