

ACDB5007: Application of Computer-Aided Engineering Processes

Computer-Aided Engineering Assignment

Peter Down
5-3-2024

Contents

Definitions, Abbreviations, and Acronyms	3
Introduction	3
Design Decisions	3
Kinematics Assessment.....	6
Simulation	7
Description.....	7
SolidWorks Static Structural Assessment	7
ANSYS Static Structural Assessment	9
Limitations of Computer-Aided Engineering	12
Conclusions	13
References	14
Appendix A.....	15

Definitions, Abbreviations, and Acronyms

Term	Definition
CAE	Computer Aided Engineering
CAD	Computer-Aided Design
Motor	Rocket Motor
FEA	Finite Element Analysis

Introduction

Computer Aided Engineering (CAE) tools are used widely across industries as they allow engineers to design, simulate, and analyse new products/tools before making them. By using CAE tools, the behaviour of an item can be predicted, and thus can also be optimised before a physical prototyping process. This saves time and costs by reducing the number of iterations needed.

CAE software is composed primarily of two types: modelling software and Computer Aided Design (CAD) software. CAD is used to create parts, put them together in assemblies, and create engineering drawings. Modelling software, such as Finite Element Analysis (FEA) packages, is used to simulate parts or assemblies in realistic loading conditions. The CAD software used for this assignment is SolidWorks and both SolidWorks and ANSYS are used to simulate a static structural scenario for the model.

This report will detail the model the author has designed, a kinematics assessment, simulations of this design, and a discussion of the limitations of CAE methods.

Design Decisions

No proprietary or classified information has been used within this assignment.

The author decided to model an accidental ignition of a rocket motor within an assembly stand. A rocket motor (known as a motor) undergoes a final assembly where a blastpipe and an igniter are fitted. By fitting a blastpipe, the motor can become propulsive in the event of an accidental ignition of propellant. By fitting an igniter, the risk of accidental ignition increases. Assembly stands include features that are designed to allow access to the motor while also restraining it. Without these features, an accidentally ignited motor can cause significantly more harm.

The diameter of the motor is 180 mm, a typical diameter for missiles such as Hellfire. The motor is a cylinder with forward and aft end rings. Together with the blastpipe, these act to make the model rigid. For the sake of simplicity, each part of the motor has been made to be 5 mm thick. The author did not consider the accuracy of the motor weight to be critical to the outcome of the simulation. Deformation within the motor was also not considered critical as a fully rigid motor would provide a worst-case result for the simulation. Figure 1 shows the motor.

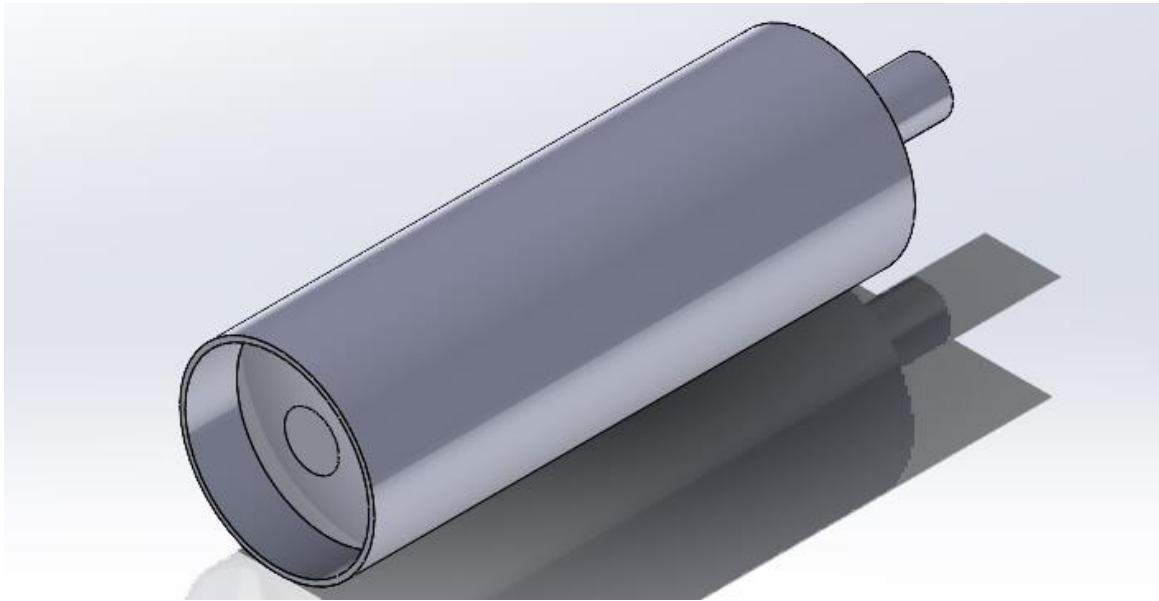


Figure 1: Rocket motor model. The forward end is left of the picture.

The assembly stand sits on a vertical 'I' shaped beam. The thickness of the material has arbitrarily been set at 20mm and the height of the beam is set at 1100 mm to put the motor at a comfortable working height [1]. The beam has a 20mm thick top plate that has bolt holes to interface with an assembly rig. Figure 2 shows this part.

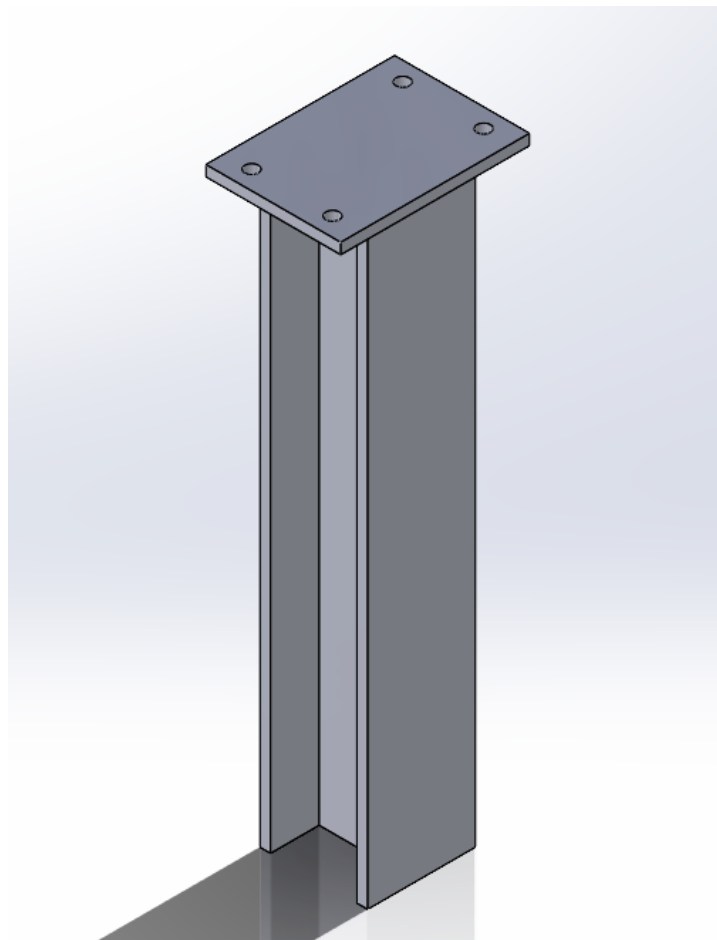


Figure 2: Vertical beam.

The assembly rig consists of a base plate, two vee blocks with tie-down arms, and a forward stop. The base plate is 30 mm thick and long enough to extend beyond the ends of the rocket motor when mounted. The additional surface

area allows for a working surface and to catch motor components should they be dropped. The vee blocks support the weight of the motor and centre it laterally to the stand. The tie-down arms are used to prevent the motor from rotating during assembly operations and to help control it in the event of accidental ignition. The vee blocks and the tie-down arms have been modelled together as a piece which surrounds and interfaces with the outer diameter of the motor. This decision was made to reduce complexity within the design. It was not considered to affect the outcome of the simulations. The forward stop interfaces with the forward end of the motor – this includes the face and the side. The purpose of this part is to prevent the escape of an accidentally ignited motor. As such, it has been made 40 mm thick. The model for the assembly rig is shown in Figure 3.

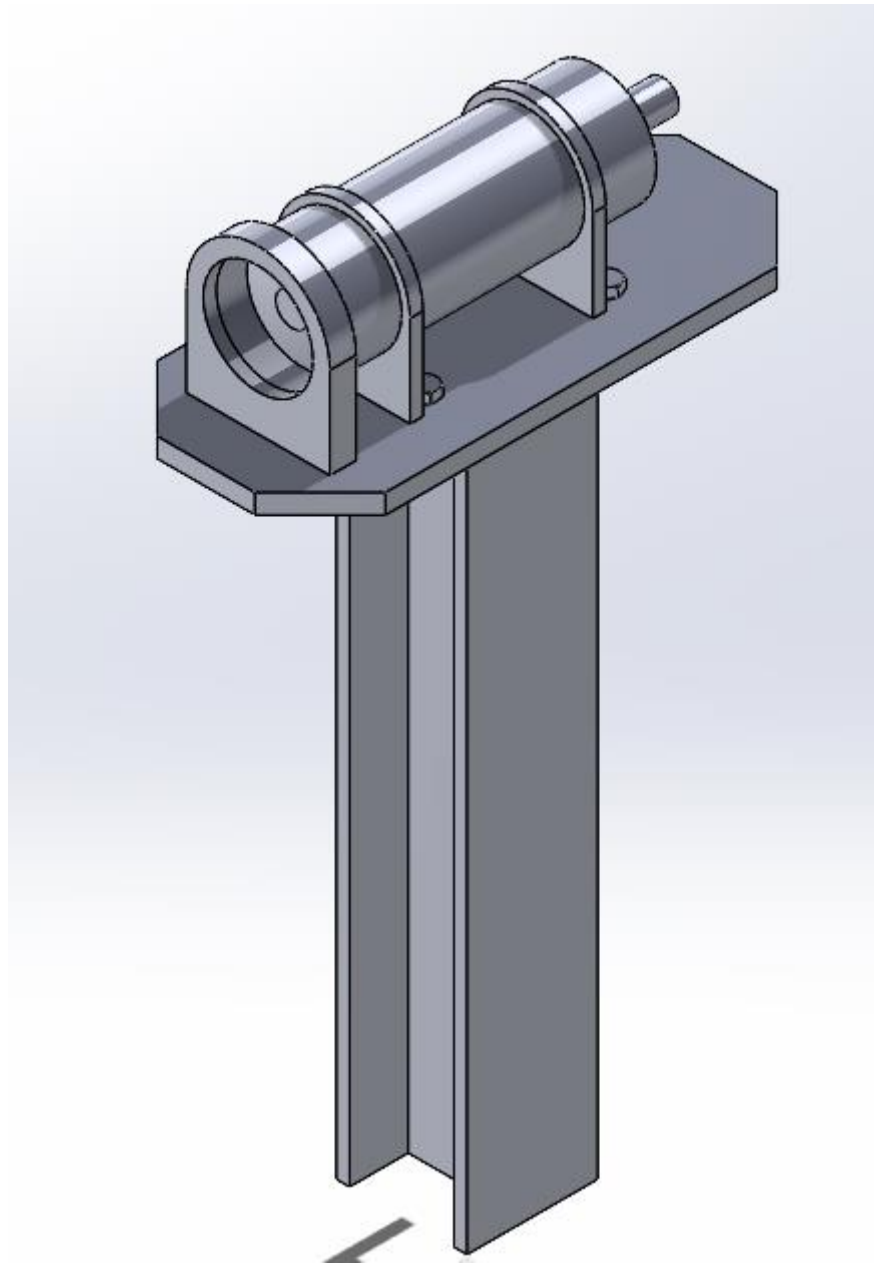


Figure 3: Assembly rig.

The load bearing components within this assembly are the motor itself, the forward stop, the assembly rig base, and the assembly stand. To make the results more realistic, these are related to bolts. The forward stop is attached to the assembly rig base with three M16 x 60 bolts [2]. The holes within the forward stop are threaded and the holes in the base are clearance holes [3]. The bolt chosen for this has a shank that is 22 mm long. The base and the vertical beam are held together by M24 x 80 bolts [4]. The holes in both components are clearance holes and nuts are used

[5]. The shank length for these bolts is 26 mm. The bolts are grade 12.9 high tensile steel – this decision was made as they must withstand high shear loadings. The M24 nuts were grade A4 maritime stainless steel.

Except for the nuts and bolts, all components are made of 304 stainless steel. This material was chosen as it is commonly available, corrosion resistant, and has an ultimate tensile strength of 505 MPa [6]. The drawings for all components and assemblies have been included in Appendix A.

Kinematics Assessment

To design an appropriate assembly stand, the thrust of the motor must be known. To calculate this, the following equation can be used:

$$F = ma$$

The mass of a Hellfire missile is 49 kg [7]. It can be assumed that the motor will make up about half of that weight and will therefore be about 25 kg. The initial acceleration of the Hellfire is about 10G and thus about 91 m/s² [8]. The thrust therefore is:

$$F = 25 * 91$$

$$F = 2275 \text{ N}$$

By designing the assembly stand to withstand an applied force of 5000 N, a safety factor of 2.2 is achieved. This is sufficient to account for the variability of thrust. The force is applied to the aft face of the motor blastpipe to simulate the thrust generated there by accelerated gasses.

The base of the assembly stand is modelled as a fixed surface. The stand would extend deep into the ground to ensure that it is well-supported. For this model however, stresses would concentrate on the stand at ground level and thus by fixing the base of the stand at ground level, real forces can be approximated.

The forward stop receives the force from the motor directly. This is attached to the rig with three M16 x 60 bolts. The total shear force of these bolts must surpass the force that is exerted on them. The first step to working out the shear force of an M16 bolt is to determine the shear area using the formula:

$$A_s = \frac{\pi d^2}{4}$$

$$A_s = \frac{\pi(16 \times 10^{-3})^2}{4}$$

$$A_s = 0.201 \times 10^{-3} \text{ m}^2$$

The shear strength of a grade 12.9 bolt is approximately 517 MPa [9]. The following formula is used to determine shear force:

$$Force = \tau \times A_s$$

$$Force = (517 \times 10^6)(0.201 \times 10^{-3})$$

$$Force = 103,917 \text{ N}$$

Therefore, one M16 bolt is sufficient to withstand the force of the motor. Using three bolts allows for redundancy in case one of the bolts is damaged and fails.

The deformation most likely to be experienced is a forward pivot of the vertical beam as this is the least robust component within the assembly. The area of greatest stress is likely to be the base of the vertical beam around the fixed surface.

Simulation

Description

The above-described design was simulated within both SolidWorks and ANSYS. Local interactions were created for all connected parts. Most of these used a bonded interaction, however some used a contact interaction.

SolidWorks Static Structural Assessment

As this simulation is a simple static structural assessment, a fine mesh was used. This was applied to all parts except the motor and can be seen in Figure 4.

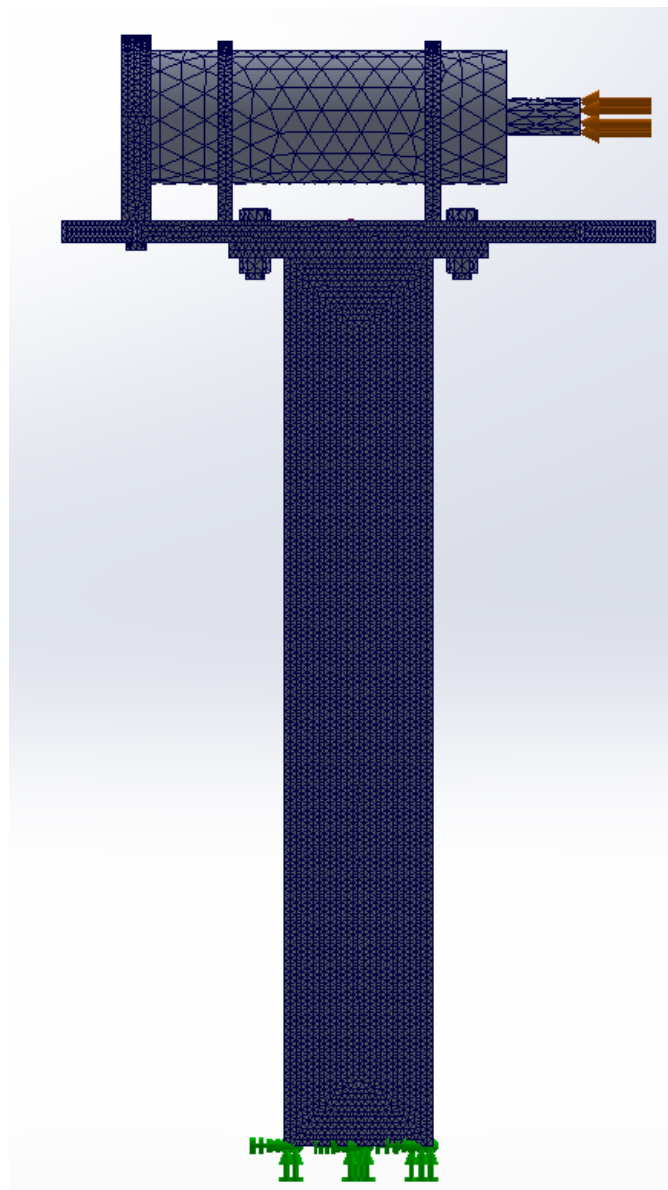


Figure 4: Mesh for static structural simulation.

The advantage of using such a fine mesh is that it allows for a more accurate understanding of the stresses present within each element. Consequently, the values for stress given by the FEA solver are also likely to be more accurate.

The drawback of finer elements is that the calculations performed by the solver are more complex. This leads to longer solving times and a higher likelihood of failure.

Figure 5 shows the deformation of the assembly stand:

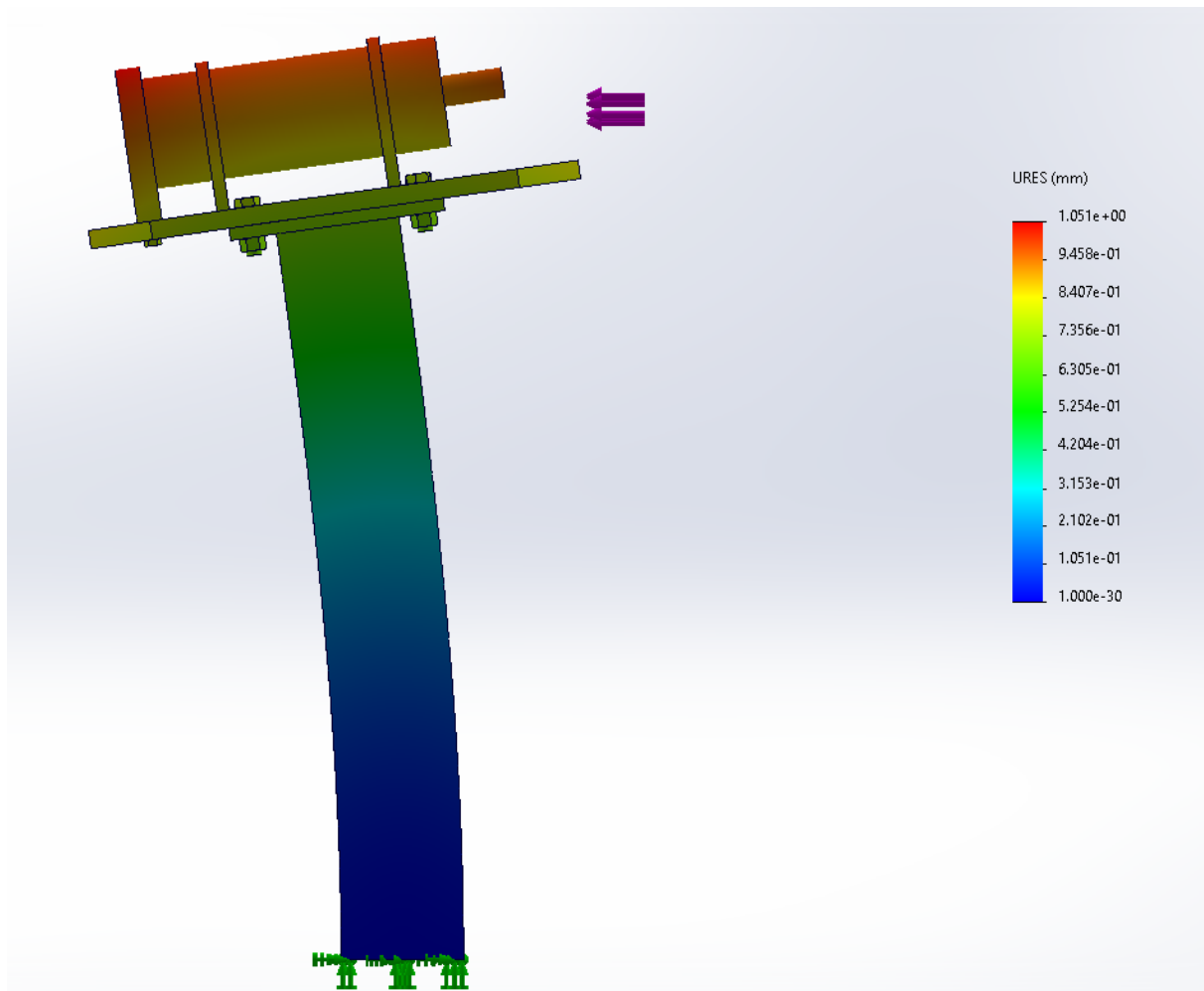


Figure 5: Deformation of the assembly stand in mm.

The maximum deformation of the stand is approximately one millimetre at the forward stop. As can be expected, the maximum deformation is furthest away from the fixed surface and furthest from the force in the direction it is acting. The visualiser presents an exaggerated view of the deformation.

As predicted, it is the vertical beam that undergoes strain and bends. This is because it is the least robust part of the assembly, being made of material that is only 20 mm thick. Furthermore, the force is acting on the opposite end of the vertical beam to where the fixed surface is and thus a large moment is experienced. The assembly rig itself remains rigid and retains its shape.

Figure 6 shows the stress within the assembly stand:

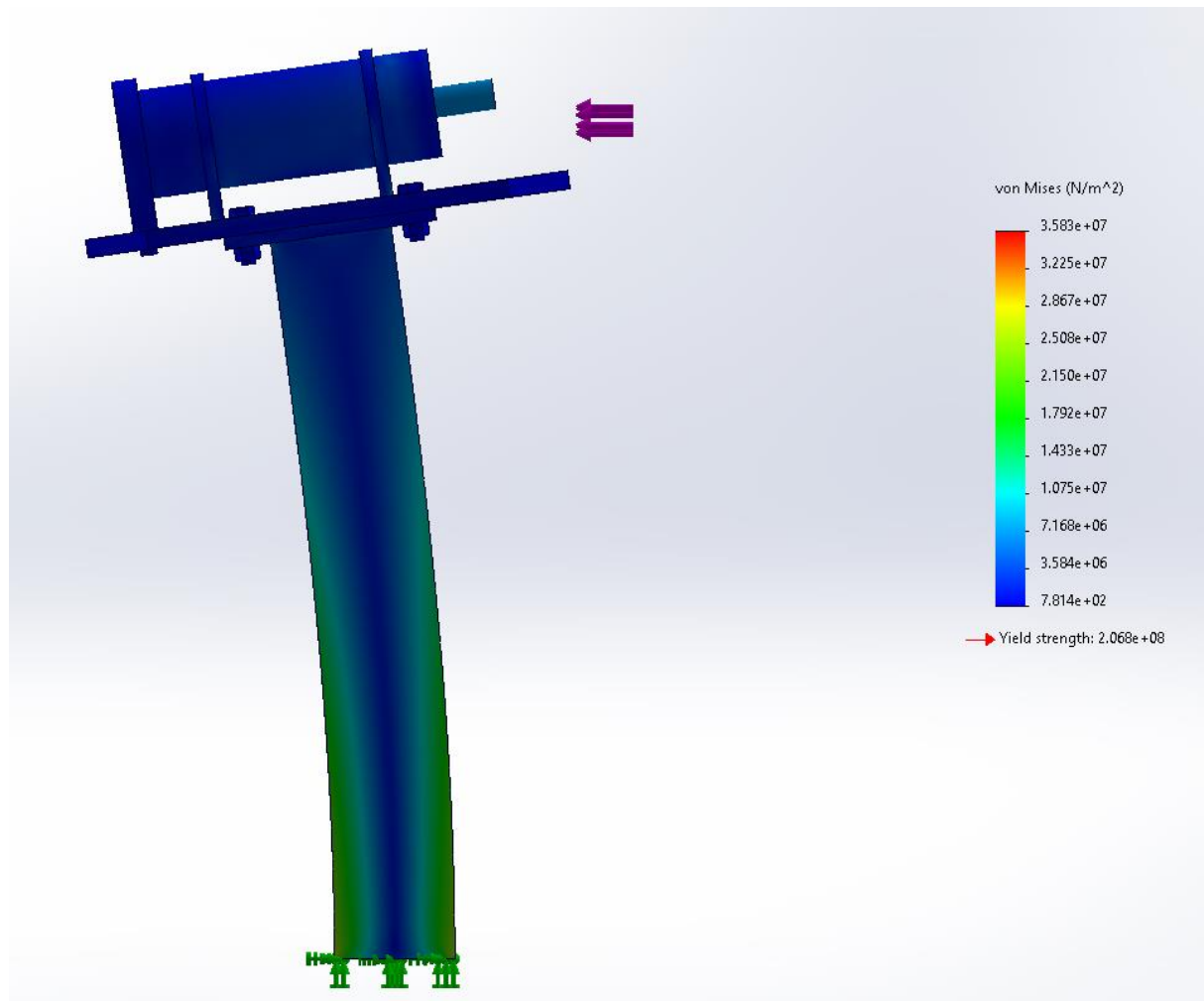


Figure 6: Von Mises stress within the assembly stand.

The maximum stress is experienced within the vertical beam at the fixed face. It is both along the forward side and the aft side of the vertical beam. The forward side is experiencing compressive stress due to the direction of the force whereas the aft side of the vertical beam is experiencing tensile stress. If the beam were stressed to the point of failure, it would fail on the aft side. The maximum stress experienced is $35.8 \times 10^6 \text{ Nm}^{-2}$. This is an order of magnitude smaller than the yield strength of 308 stainless steel.

ANSYS Static Structural Assessment

A similar mesh was used for the ANSYS model, see Figure 7.

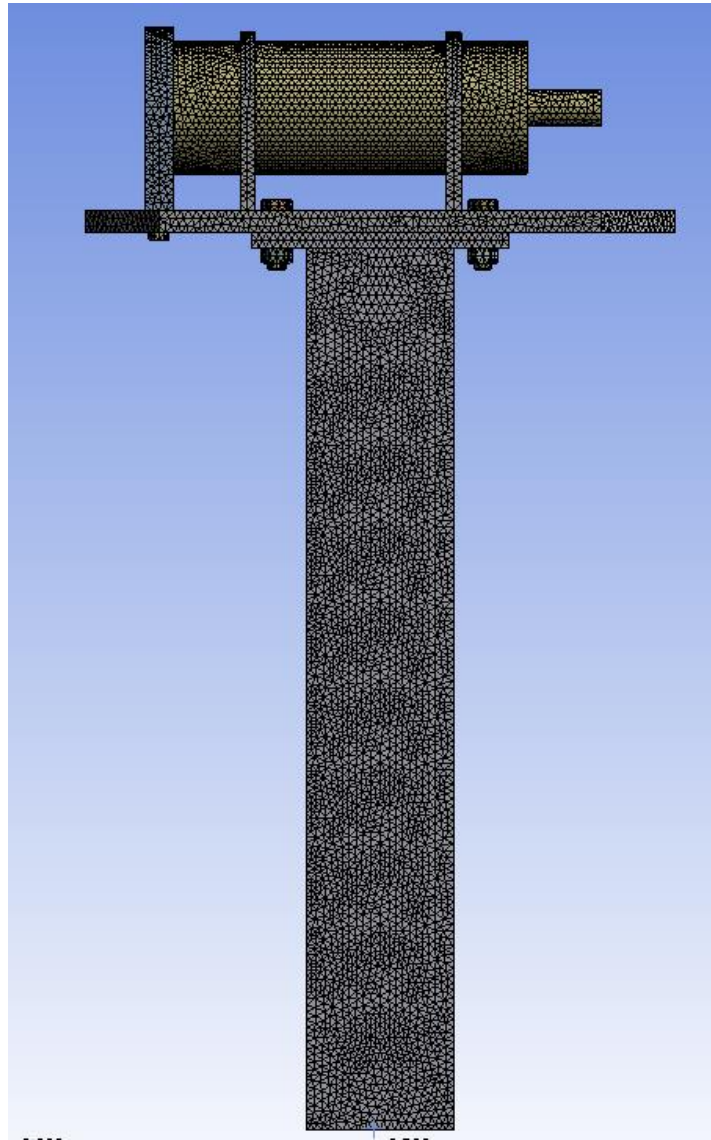


Figure 7: Mesh for static structural simulation.

This mesh is like that used by the SolidWorks simulation. It is also applied across all parts including the motor.

Figure 8 shows the deformation within the ANSYS solver:

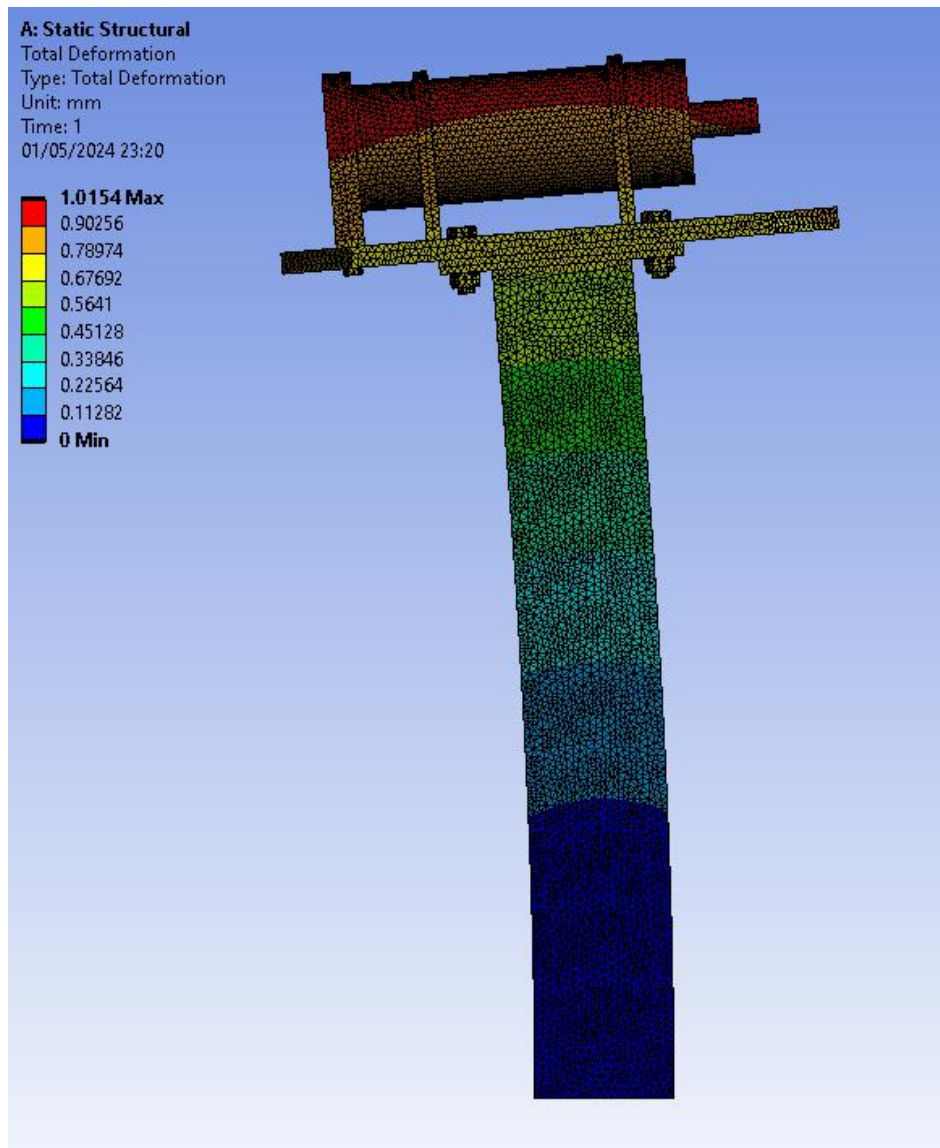


Figure 8: Deformation of the assembly stand in mm.

The deformation in ANSYS is like that in SolidWorks being only 0.04 millimetres under and thus validates this value. The same pattern is seen with the deformation occurring primarily at the top of the assembly due to the fixed surface on the base of the vertical beam. Bending occurs in the same place on the beam.

Figure 9 shows the stress within the assembly stand:

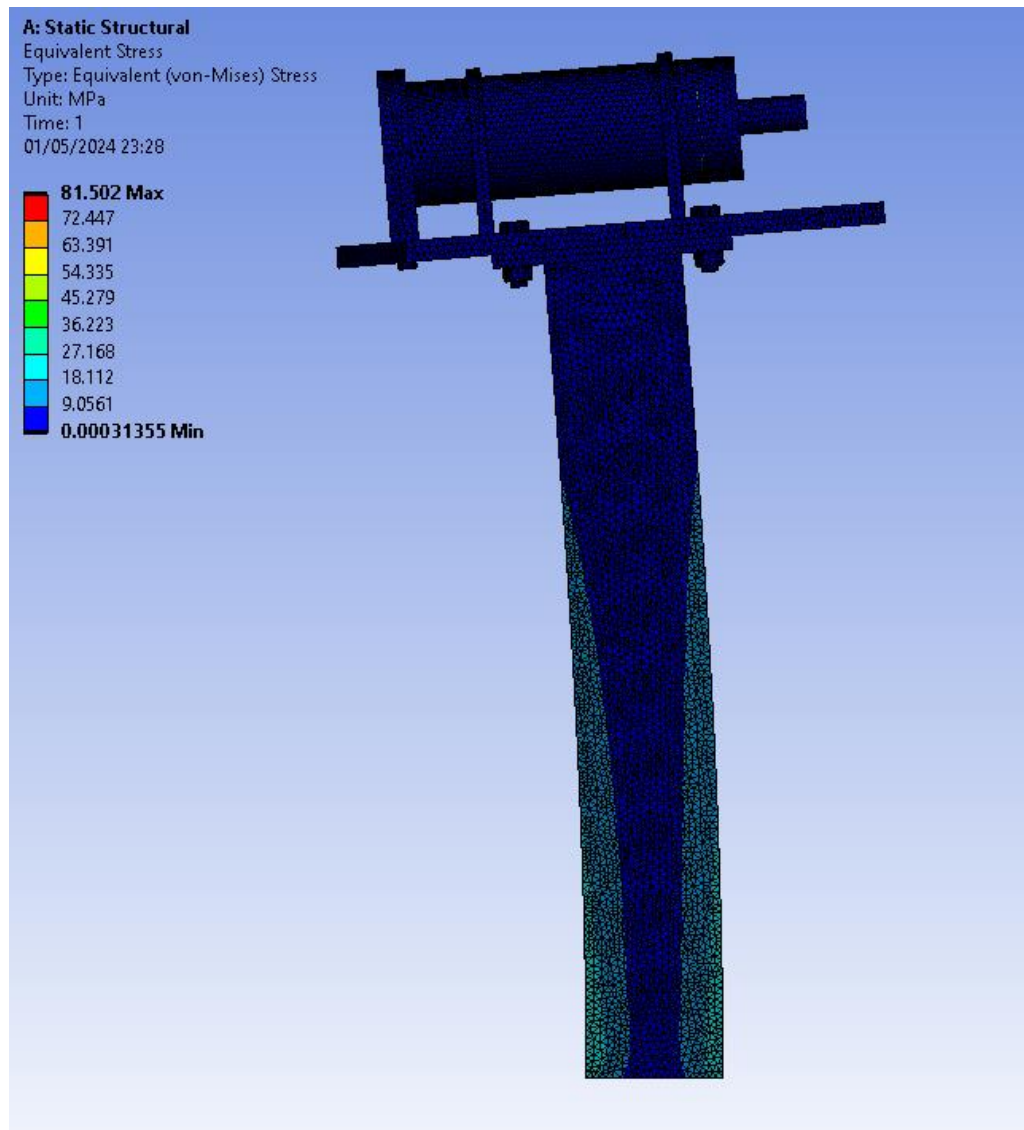


Figure 9: Von Mises stress within the assembly stand.

The greatest stress occurs at the base of the vertical beam due to the fixed surface again. The maximum stress experienced is 81.5×10^6 Pa. One Pascal is equivalent to one Newton per metre squared (the unit used in SolidWorks). As such, the stress calculated in ANSYS is slightly more than double that calculated within SolidWorks. It is unclear why this is the case; however, it likely has to do with how they perform calculations around a fixed surface.

While these results do not validate each other, they show that even if the maximum stress is significantly higher, the assembly stand still performs its function adequately as the rig is rigid and not deformed.

Limitations of Computer-Aided Engineering

Despite the advantages that CAE tools provide, they also have limitations. These must be accounted for when using them to interpret results and compare them to real products or equipment.

As CAE tools are all digital, there is a risk of data loss through corruption or accidental deletion. Consequently, robust backup and data management strategies must be implemented to mitigate this risk and ensure data is protected.

FEA software requires significant processing power and can take a long time to perform analyses. This is compounded by the risk of crashes partway through simulations. The result of this is that it can take a long time to progress through iterations in design.

Poor meshes within FEA software can also lead to issues. The quality of the results can depend on the size of the mesh and the shape of the elements. It can often be difficult to know how to make a successful mesh to get accurate results when approaching a task. A bad mesh can also lead to the software crashing. While the results are saved at the point of the crash, the remainder of the simulation cannot be known without restarting the entire process which costs additional time. The difficulty of finding a good mesh for each simulation can force the adoption of a trial-and-error method.

After completion of an FEA simulation, the quality of results cannot be known until another source of data is provided that can be used to validate. As such, they work well when implemented into a larger programme that involves physical testing but cannot be used independently. Validation allows refinement of the model to bring results closer to the data from physical tests.

To use CAE tools in development or research, many assumptions must be made. This can result in potentially incorrect inputs into the model which can lead to incorrect conclusions. This could be significant during down selection activities for example, where two options have comparable properties, but the worse option is chosen due to bad inputs to the modelling software.

Another limitation of CAE tools is that they can be incapable of solving especially complicated problems. In consequence, the problem needs to be broken down into more manageable sections that can be separately modelled, and an inference can be made about the overall problem. An example of this is the difficulties of modelling the fluid flow through the nozzle of a rocket motor.

Conclusions

CAE tools are used across industries for their beneficial characteristics – namely that they can accurately simulate real problems if they are provided with accurate inputs. This is done at low cost and with a quicker turn-around than physical testing programmes.

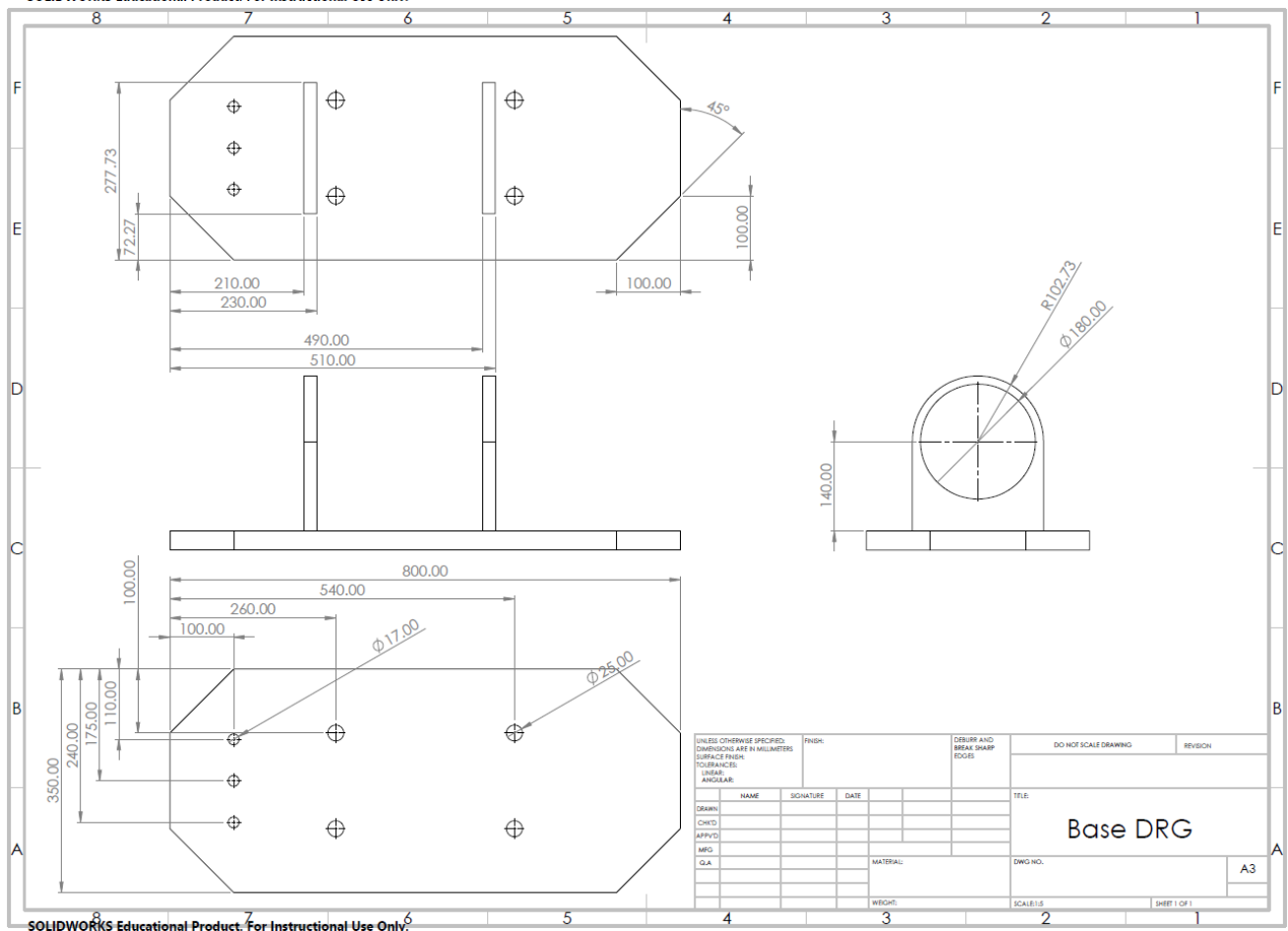
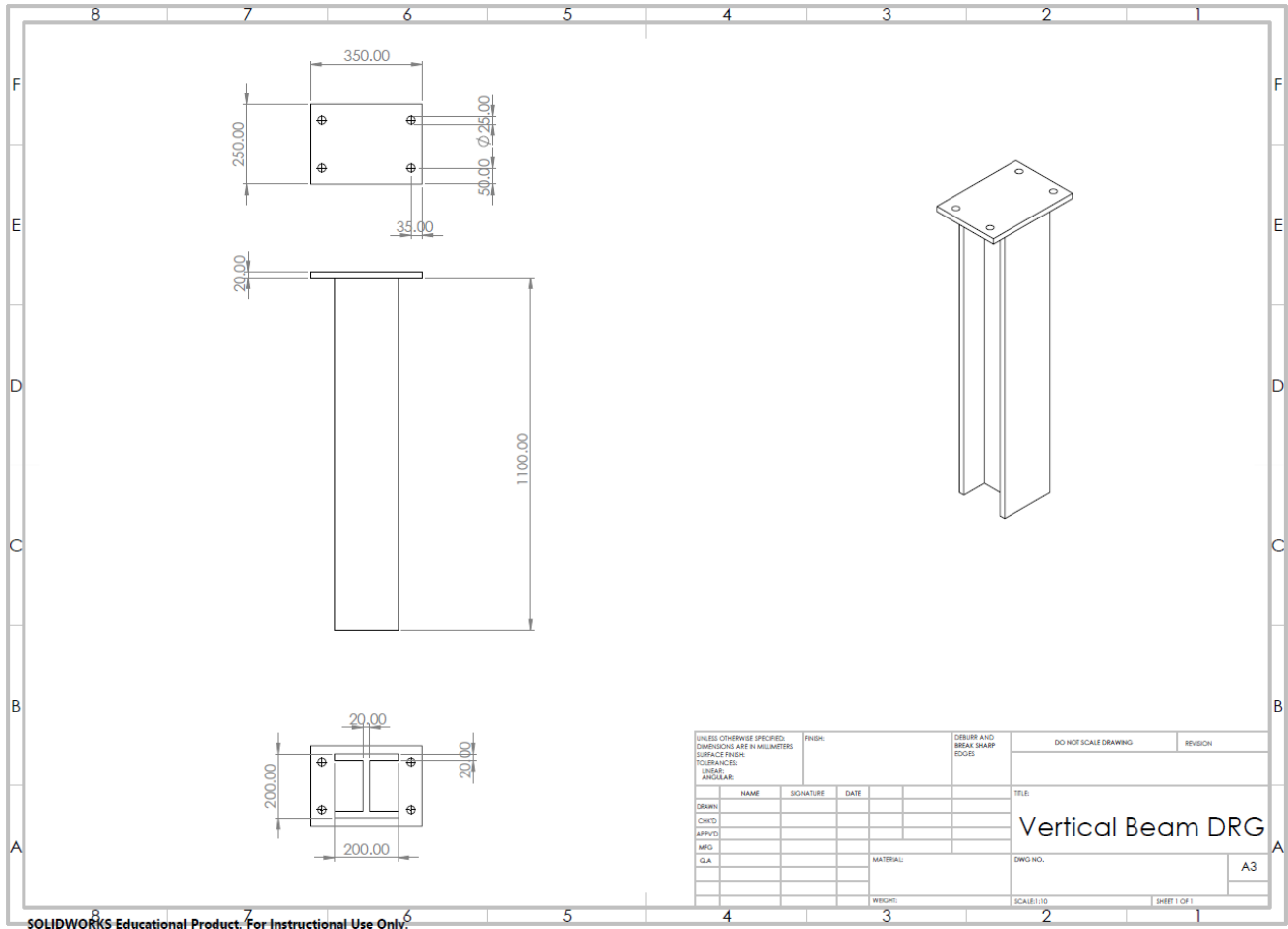
The design of an assembly stand for a rocket motor is made complex by having to withstand the thrust of the motor in the event of an accidental ignition. The assembly stand designed by the author can withstand an accidental ignition of a motor based on the Hellfire missile's motor. It does this with a maximum deformation of about one millimetre. This result is validated as it has been reproduced in two different FEA software packages.

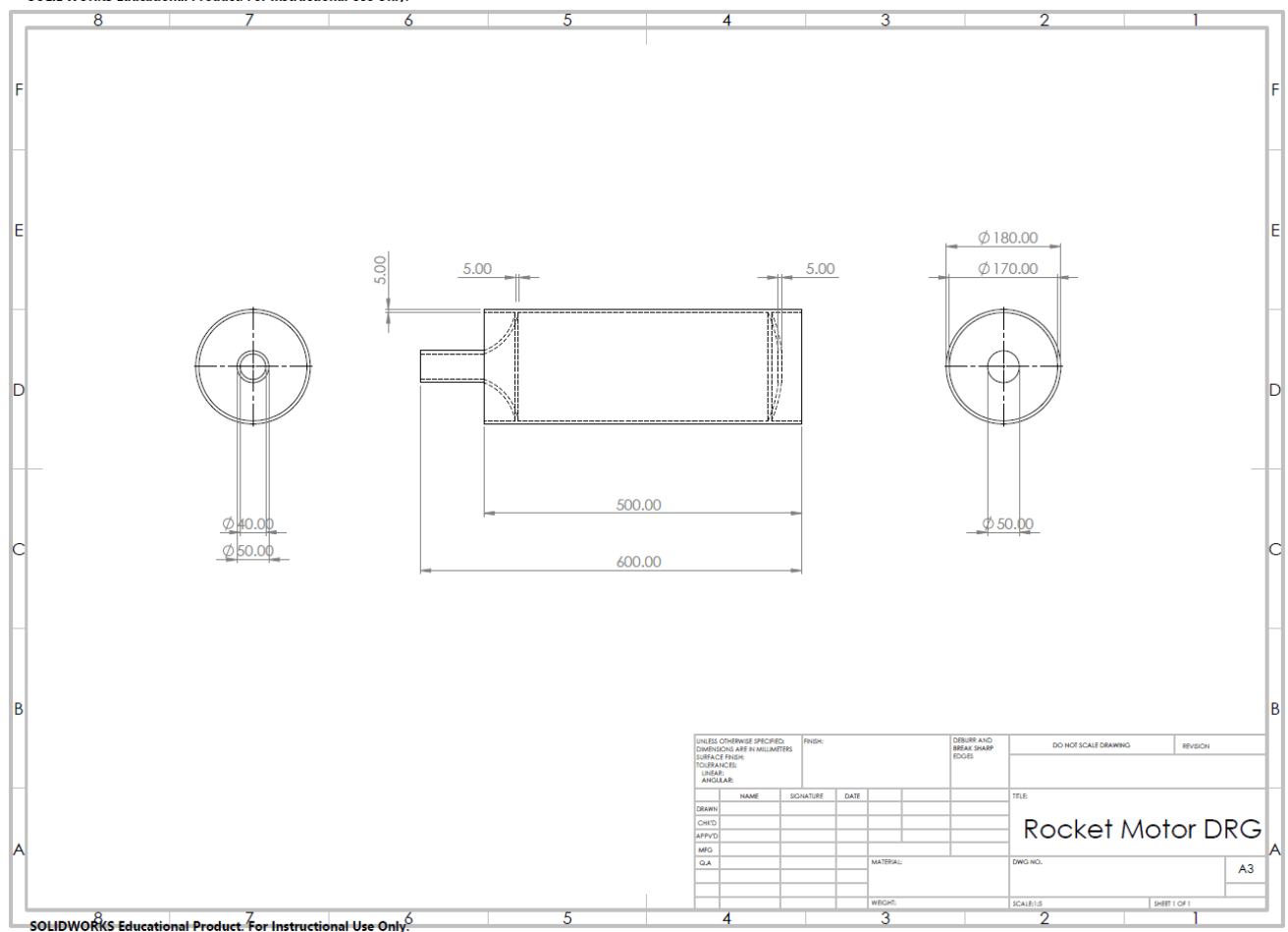
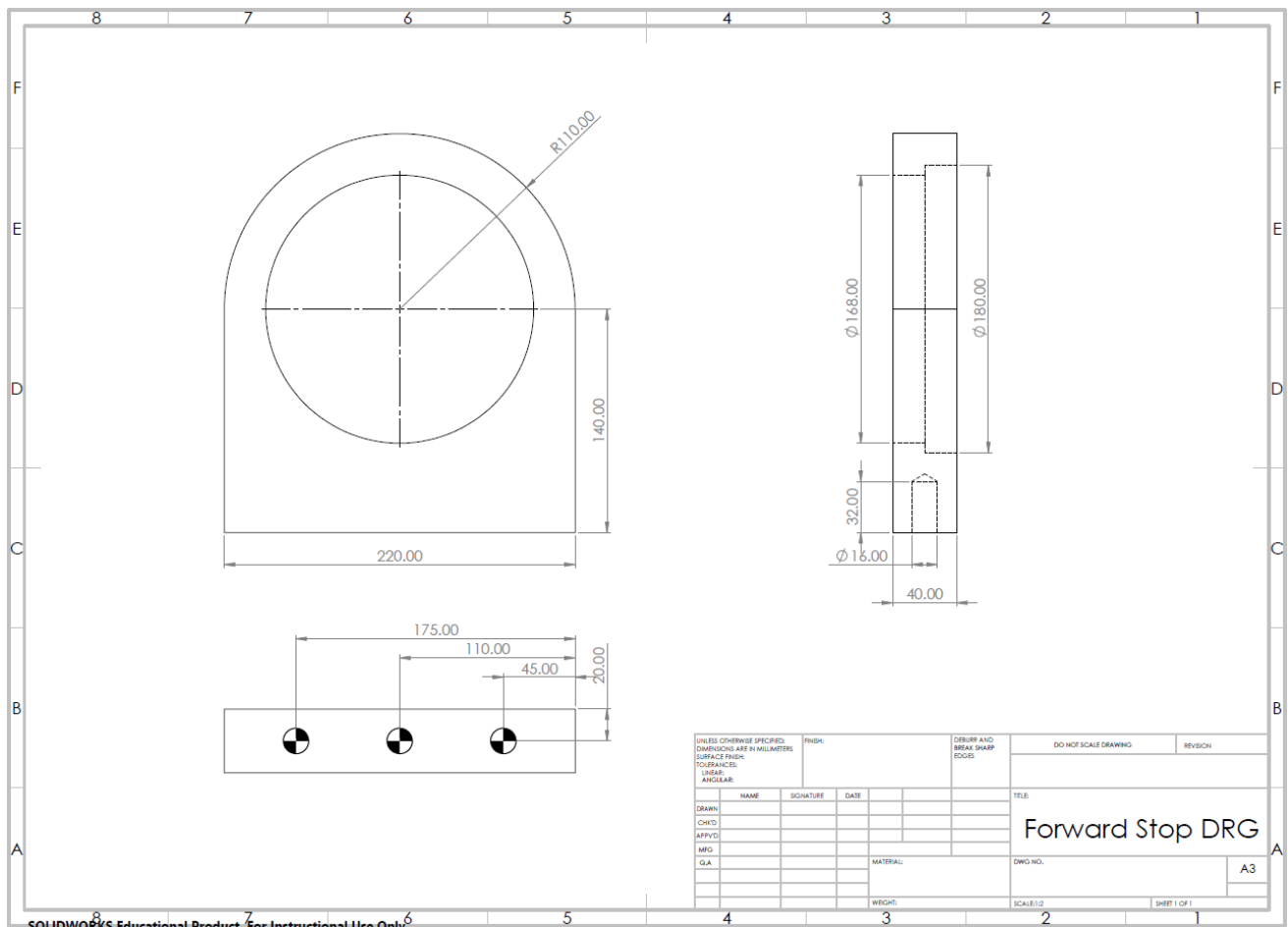
CAE tools also have several limitations. Most of these do not diminish their role when used alongside physical testing. Through numerous mechanisms, the accuracy of the data produced by CAE software can be reduced, and thus, using CAE software independently without means of verification is not a viable option.

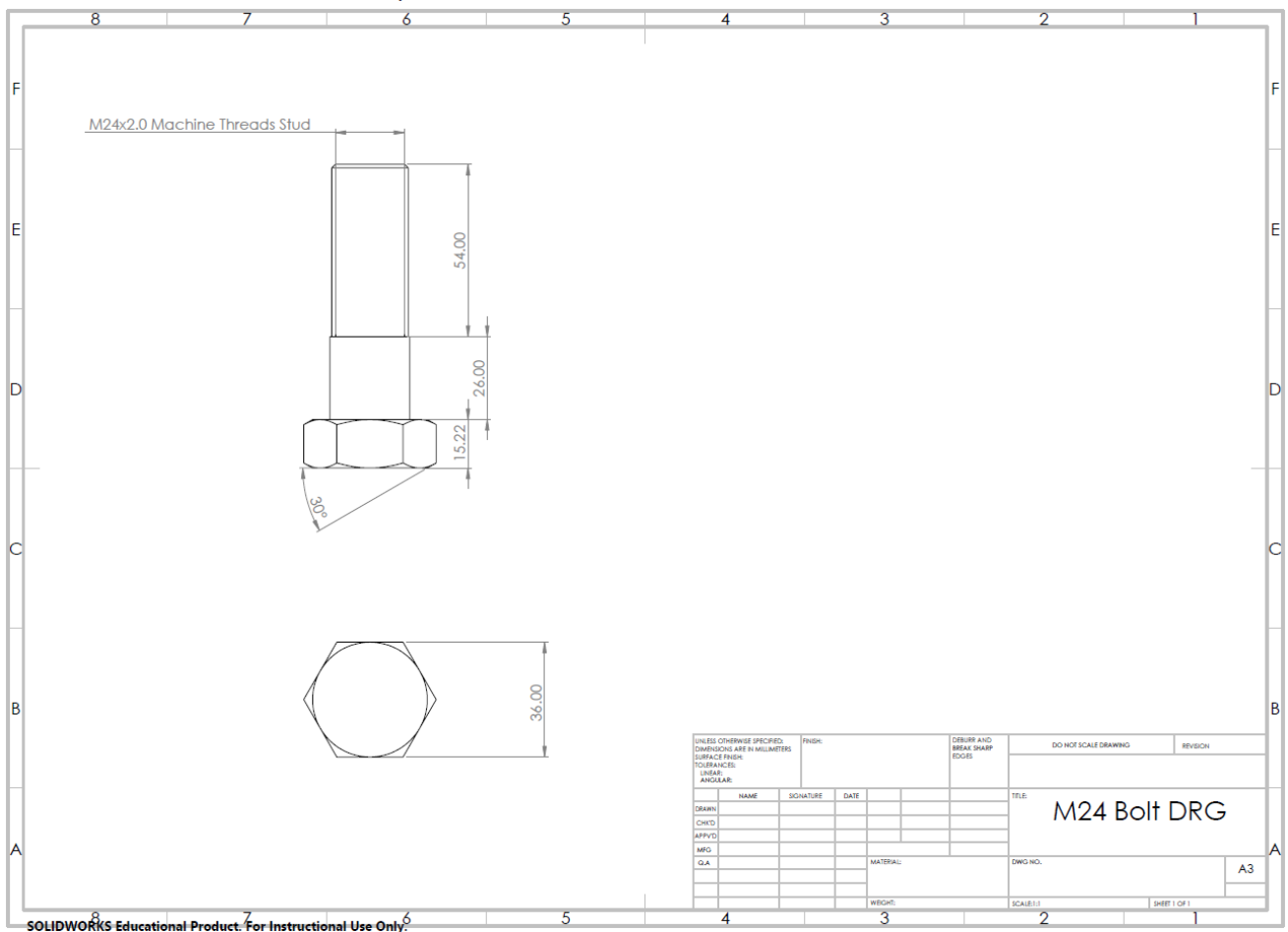
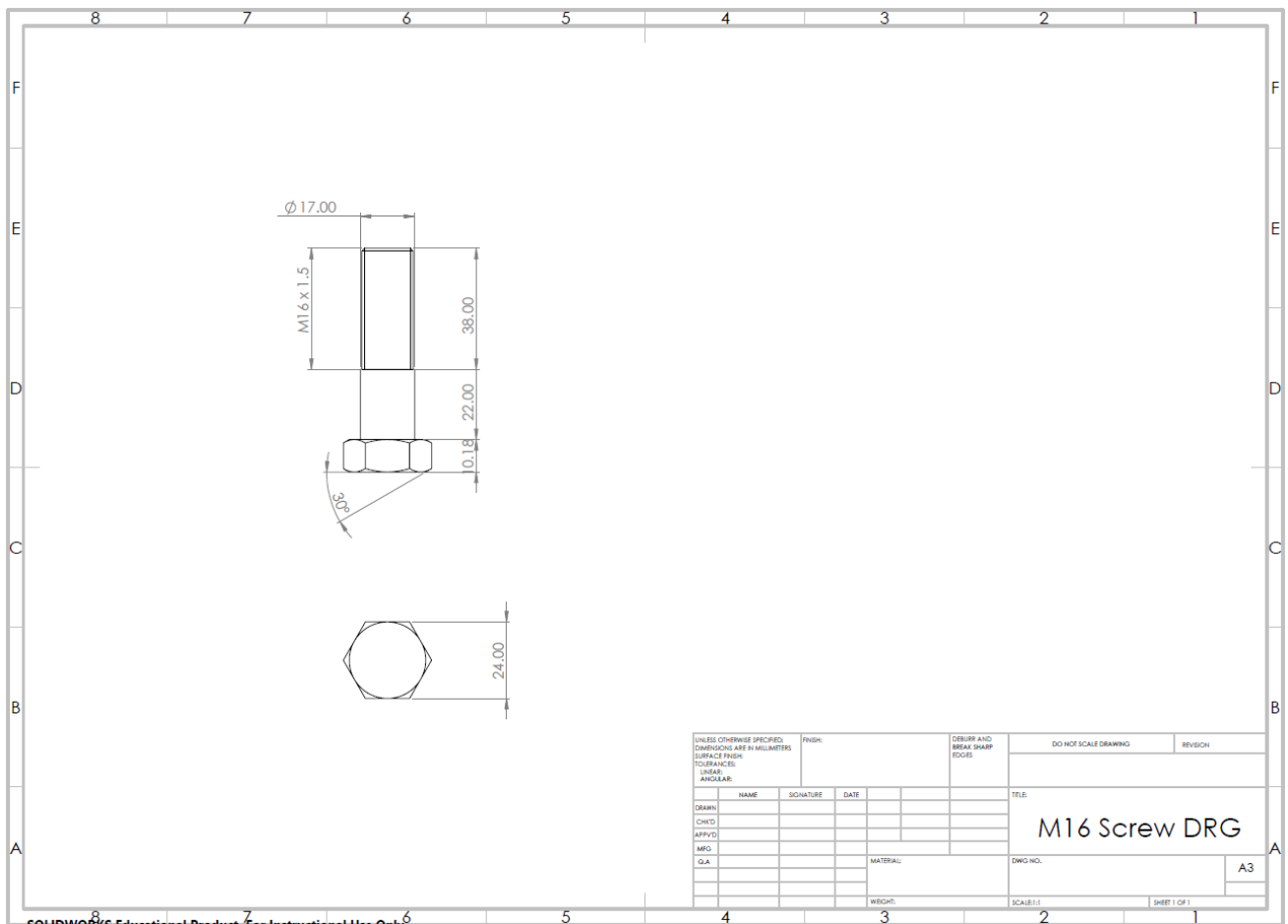
References

- [1] Ergo Desks, "What Height Should My Desk Be And Why Does It Matter?," 15 01 2020. [Online]. Available: <https://ergodesks.co.uk/blogs/news/what-height-should-my-desk-be>. [Accessed 03 05 2024].
- [2] "M16 x 60mm Plain Hexagon Bolts (DIN 931) - High Tensile Steel (12.9)," Accu, [Online]. Available: <https://www.accu.co.uk/metric-hexagon-bolts/429794-SEB-M16-60-12-9>. [Accessed 01 05 2024].
- [3] "Clearance holes for metric bolts," Stephens Gaskets Limited, [Online]. Available: <https://www.stephensgaskets.co.uk/clearance-holes>. [Accessed 01 05 2024].
- [4] "M24 x 80mm Plain Hexagon Bolts (DIN 931) - High Tensile Steel (12.9)," Accu, [Online]. Available: <https://www.accu.co.uk/metric-hexagon-bolts/429820-SEB-M24-80-12-9>. [Accessed 01 05 2024].
- [5] "M24 Hexagon Nuts (ISO 4032) - Black Marine Stainless Steel (A4)," Accu, [Online]. Available: <https://www.accu.co.uk/hexagon-nuts/406471-HPN-M24-V1-A4-BL>. [Accessed 01 05 2024].
- [6] "304 Stainless Steel," MatWeb, [Online]. Available: <https://matweb.com/search/DataSheet.aspx?MatGUID=abc4415b0f8b490387e3c922237098da&ckck=1>. [Accessed 03 05 2024].
- [7] Department of the Army, Office of the Deputy Chief of Staff for Research, Development, and Acquisition, "Longbow HELLFIRE," in *Weapons Systems*, 1998, p. 125.
- [8] Army Technology, "AGM-114 Hellfire II Missile," 21 01 2020. [Online]. Available: <https://www.army-technology.com/projects/hellfire-ii-missile/?cf-view>. [Accessed 03 05 2024].
- [9] Trident Racing Supplies, "BOLTS-COMMERCIAL GRADE," [Online]. Available: [https://tridentracing.co.uk/product/bolts-commercial-grade/#:~:text=12.9%20grade%20Socket%20Cap%20heads,75%2C000%20\(517MPa\)%20shear%20strength..](https://tridentracing.co.uk/product/bolts-commercial-grade/#:~:text=12.9%20grade%20Socket%20Cap%20heads,75%2C000%20(517MPa)%20shear%20strength..) [Accessed 03 05 2024].

Appendix A







8 7 6 5 4 3 2 1

F

E

D

C

B

A

30°

21.50

36.00

M24x2.0

UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH TOLERANCES: LINEAR ANGULAR:		FINISH:		DEBURR AND BREAK SHARP EDGES		DO NOT SCALE DRAWING		REVISION	
GRAPH:	NAME	SIGNATURE	DATE			TITLE:			
CHKD:						M24 Nut DRG			
APPVD:						DWG NO.:		A3	
MFG:						SCALE:		SHEET 1 OF 1	
Q.A.						WEIGHT:			

SOLIDWORKS Educational Product. For Instructional Use Only.