

Computer Aided Engineering

ACDB5007

Jacob Smith

Table of Contents

Content	Section
Introduction	1
Computer Aided Engineering	2
Design requirements	3
Initial design	4
Validation	5
Validation results	6
Design simulation	7
Simulation results	8
Design improvements	9
Design evaluation	10
Conclusion	11
References	12
Appendix	13

Introduction

This report has been produced for the purpose of validating a real-world system designed in a Computer Aided Engineering (CAE) programme and tested in a computational simulation environment.

The modelling within this report will be produced using solidworks. Solidworks is a programme produced by Dassault Systems a French software company [1]. Solidworks is an easy to use an intuitive software allowing the user to achieve the desired results. The modelling software that will be used in this report will be Ansys modelling. Ansys is an analysing software that provides the user with various simulation possibilities such as autonomous sensors, battery simulation, 3D designs, Multiphysics, antenna design and fluid dynamics, with countless other applications [2]. Due to the nature of the item in question the simulations within this report will be based around static loading with a focus being on the displacement (bending), stress and strain experienced by the model in the simulation.

Great effort will be taken to validate the simulation modelling by completing a validation exercise against a known model design. This validation effort will allow the results of modelling the equipment to be taken as true to life and representative of the real-world system.

For the purpose of this report the real-world system that will be modelled will be based around a requirement for a weapon system launcher to be placed on a stand for maintenance and processing before being loaded onto a platform. The overall requirements for this design will be summaries in a future chapter. Due to the weapon system in question being a real-world item which characteristics are protected under the Government Security Classifications Policy [3] and the nature of this report being unclassified. All dimensions, weights, specifications, and requirements contained within this document are not representative of the real-world equipment and any similarities are purely coincidental.

Computer Aided Engineering

Computer Aided Engineering (CAE) has modernised the engineering design process, allowing engineers access to tools for simulation, analysis, and optimisation allowing for greater efficiencies during the design process of a new project. These tools have allowed engineers to predict the performance and identify potential flaws in their designs before physical prototypes are constructed, saving time and resources. Despite this their considerable utility, CAE processes come with inherent limitations and simplifications that must be considered to ensure an effective output as part of the design process.

One of the primary limitations of CAE is the dependence on the quality and accuracy of the input data. The output of a CAE process is only as reliable as the input data. This includes material properties, boundary conditions, and loading conditions, which must be accurately represented in the model. However, obtaining precise input data can be challenging, and assumptions often need to be made, potentially introducing errors into the simulation results [4].

CAE tools often require simplifications of complex real-world physics due to the limitations present on everyday computer systems. For example, in simulating fluid dynamics or stress distribution, engineers might assume laminar flow or linear elastic material behaviour, ignoring turbulence or plastic deformation. While these simplifications can allow a basic computer system to undertake these simulations, they can also lead to discrepancies between the simulated results and real-world behaviour, especially under extreme conditions or in scenarios where the simplified inputs play a significant role. Another consideration is the computational cost associated with high-fidelity CAE simulations. Detailed models that account for complex interactions and phenomena require significant computational resources and time, which can be prohibitive for some businesses. This requires a balance between model detail (accuracy) and computational feasibility, often leading to further simplifications. [5].

Interpretation of CAE results requires a deep understanding of both the underlying physical principles and the specific CAE tool being used. Misinterpretation of the results can lead to incorrect conclusions, potentially compromising the design's integrity. The limitations and required simplifications inherent in CAE processes highlight the need for these tools to be used as part of a comprehensive design and validation process, not as standalone solutions. CAE should inform and guide the engineering design process, supplemented by empirical testing and validation to confirm the performance of the final design. It is crucial for

engineers to recognise the limitations of their simulations, understand the potential sources of error, and apply their judgement in interpreting the results within the context of the broader design project [6].

In conclusion, while CAE offers powerful capabilities for enhancing the engineering design process, a critical understanding of its limitations and the simplifications involved is essential. By correctly integrating CAE tools with traditional engineering practices and validation techniques, engineers can use these technologies to their full potential, leading to innovative designs that are both efficient and reliable.

Design requirements.

As with any engineering project there are a list of requirements. Requirements are the specific standards, factors, or conditions a project or design must meet in order to be deemed successful. These requirements allow the designer to understand what the equipment or project goals are, the limitations there might be and what they wish to achieve [7].

The item to be modelled must be designed to act as a stand (hence forth known as Stand) for an item of equipment to be worked on when not attached to a platform or contained within its storage container. The requirements surrounding the stands design are as follows:

1. The stand must be able to be mounted to an MHU-191 ammunition trolley.
2. The stand must be able to accommodate the dimensions of the equipment in question. *
3. The stand must be able to hold a weight of 200kg's** with no more than 1.5mm of displacement in its structure.
4. The stand must spread the weight over an area 5000 cm^2 .*.
5. The stand must keep the equipment secured upon itself with no additional lashings.
6. The stand must allow for ease of access to the top of the equipment at a height of no less than 1 meter above the ground.
7. The stand must be constructed with the use of the minimum number of parts possible to achieve the other stipulated requirements.
8. The stand must be capable of being assembled with basic hand tools.
9. The stand must be constructed of Aluminium alloy 6061 T6.

*The exact dimensions are classified and omitted from this report, all dimensions mentioned are not representative of the real-world equipment and are purely fictional.

**This weight is not a representative weight of the real-world equipment and is used solely as a simulation target.

Initial design

The initial design consists of three parts with two parts being duplicated for simplicity, pictured below in figure 1. The two "T" shaped vinticle supports (Going forward will be known as T-Mounts) are identical allowing for simplified manufacturing process. The bottoms of the T-mounts have been designed to allow mounting onto the MHU-191 trolley

with its two different widths of mounting rails. This design allows for either of the T-mounts to be mounted in any of the mounting orientations without effecting the assembly of the stand, therefor simplifying the design.

The two T-mounts are connected with a single beam using a mortis and tenon joint, which provides a simplistic and solid mounting option [8]. This assembly method relays on the T-mounting bolts used to secure the T-mounts to the trolley as the sole part of the assembly requiring hand tools.

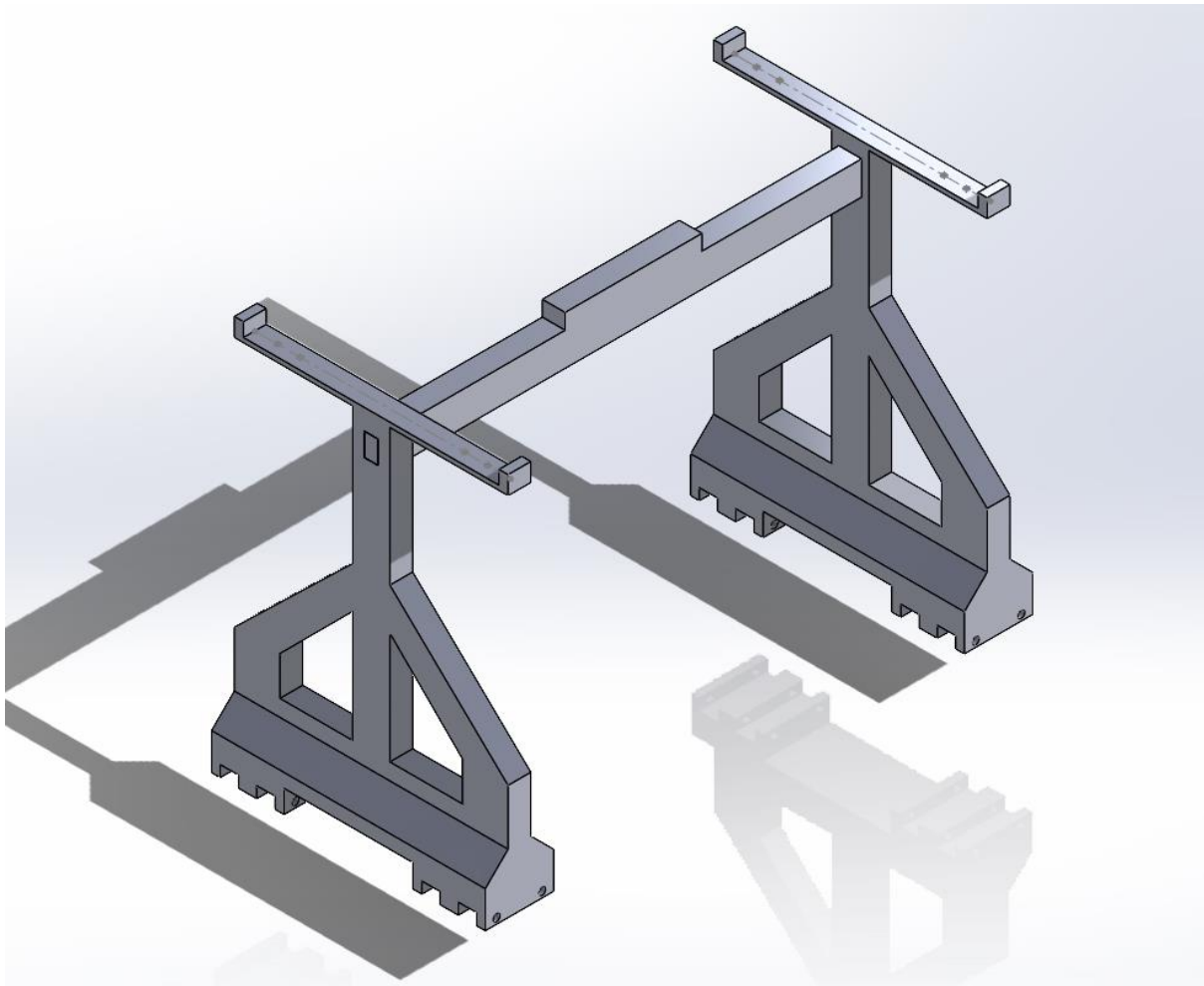


Figure 1: Design variant 1

The two end vertical lips on the T-mounts are to secure the equipment onto the mounting surface preventing it from slipping off the ends. The surface area the equipment mounts to on the three top surfaces spreads the equipment's weight over 7100cm^2 (highlighted in figure 2 below with the red circles), calculations found in Appendix A. With this in mind it can be argued that current design meets all the requirements stated above, with the exception of requirements 3 and 9. These two requirements compliance will be demonstrated during the simulation section of this report.

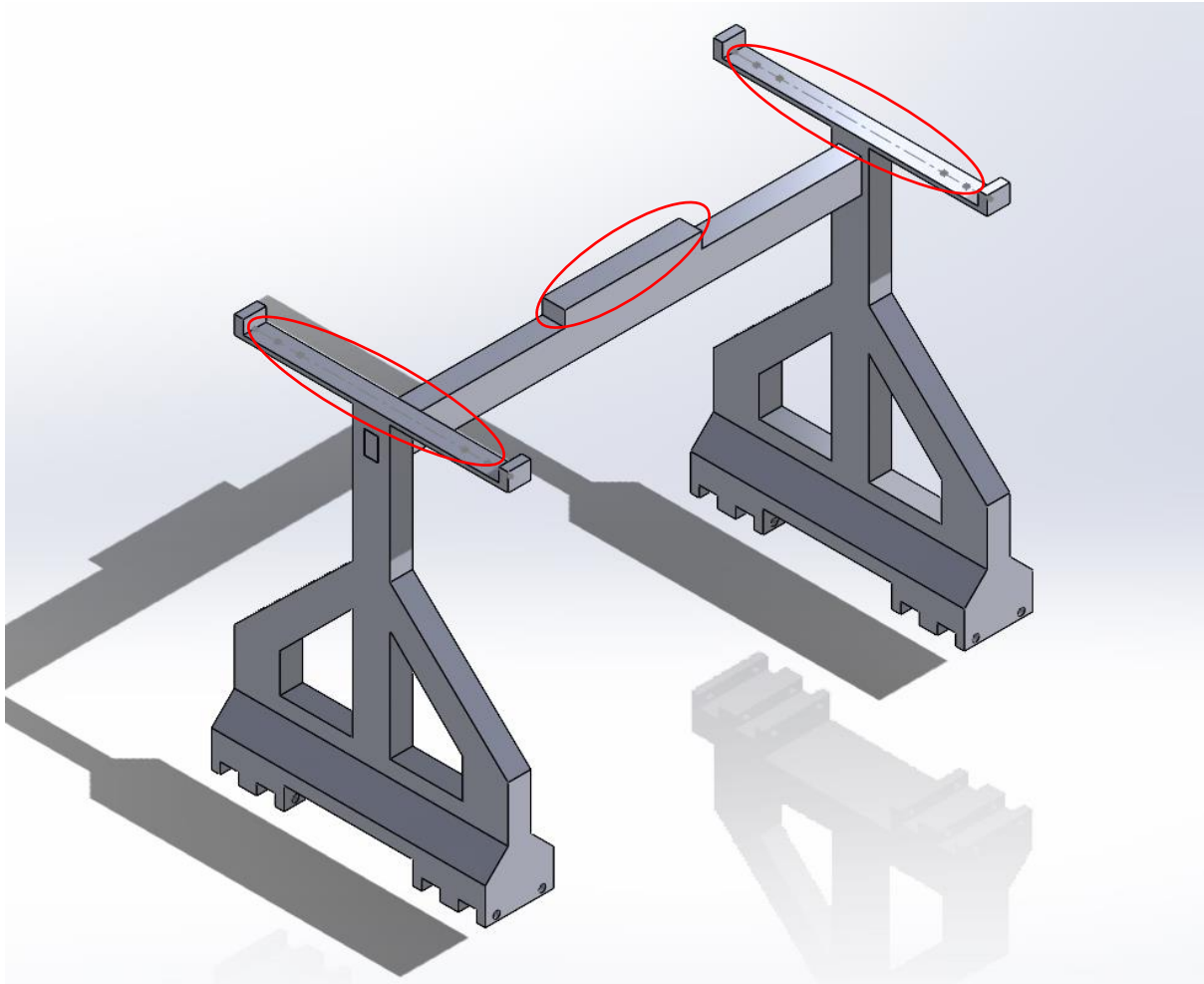


Figure 2: Highlighted load bearing areas.

Validation

Simulation validation is a cornerstone of CAE as it ensures that the simulation models accurately represent real-world scenarios. This validation process is critical for several reasons.

Accuracy of Predictions: The primary purpose of CAE is to predict the behaviour of designs under various conditions. Validation confirms that these predictions closely align with actual outcomes, ensuring that engineers can trust the data provided by their simulations when making design decisions [9].

Cost Efficiency: Physical prototyping and testing can be expensive and time-consuming, especially for complex systems. By validating simulations against real-world results, companies can reduce the number of physical prototypes, saving both time and money [10].

Risk Mitigation: In industries like aerospace, automotive, and defence, design failures can have catastrophic consequences. Validation of simulations helps identify potential issues before products are built and deployed, significantly reducing the risk of failure, and ensuring safety and compliance with industry standards [11].

Improved Understanding of Models: The process of validation often reveals discrepancies between the simulated and real-world results. Investigating these discrepancies can lead to a better understanding of the underlying physics and the limitations of the simulation, driving improvements in both the models and the design process [9].

Confidence in Decision Making: Validation provides engineers and decision-makers with confidence that their designs will perform as expected in the real world. This confidence is crucial for making informed decisions about materials, design modifications, and investments in further development [12].

Innovation and Optimisation: Validated models allow for more aggressive innovation and optimisation of designs. Engineers can explore a wider range of scenarios and design parameters with confidence, pushing the boundaries of performance and efficiency [9].

In summary, simulation validation in CAE is vital for ensuring the reliability and accuracy of simulations, which in turn supports cost-effective, safe, and innovative engineering design processes. Without validation, the utility of CAE simulations would be significantly diminished, as their predictions could not be trusted to inform critical design and manufacturing decisions.

As part of the validation effort for the simulation contained within this report, validation simulations were carried out across three different aspects closely relating to the design being simulated.

A beam bending validation exercise was undertaken to demonstrate the simulation of the whole stand was valid and representative of the real-world environment. To undertake this simulation the validation effort used four different simulations and mathematical equations. A beam of similar dimensions and using the same material properties to the horizontal beam shown in figure 1 & 2 above was used. The validation beam was the tested with the following methods.

1. Fixed at one end with a universal load across its length.
2. Fixed at one end with a single point load placed at the opposite end to its fixing points.
3. Fixed at both ends with a universal load across its length.
4. Changing the mesh size, 1 simulation with a fine mesh and 1 simulation with a coarse mesh.

Validation efforts 1 to 4 were completed in 2 CAE simulations, 1 with Solidworks and 2 with Ansys. Validation efforts 1 to 3 were also supplemented with mathematical equations for the beam deformation, the formulas used can be seen in figure 3 below as well Appendix A for the calculations. Validations simulations were compared using the maximum deformation seen across all simulations and calculations.

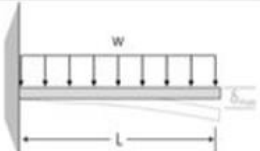
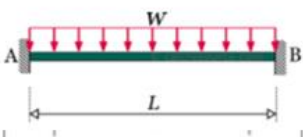
Beam and load cases	Maximum Beam Deflection
	$\delta_{max} = \frac{PL^3}{3EI}$
	$\delta_{max} = \frac{wL^4}{8EI}$
	$d_u = \frac{wL^4}{384EI}$

Figure 3: Beam validation calculations [9]

The following parameters were used as part of the validation simulations and calculations.

- Dimensions - 1000 (L) x 50 (W) x 72.5 (H) mm
- Force applied – 2000N.
- Material - aluminium alloy 6061 T6

The following material properties are contained within the two-simulation software for aluminium alloy 6061 T6:

Parameter	Solidworks	Ansys
Density	2700 Kg/m ³	2713 Kg/m ³
Poisons ratio	0.33	0.33
Ultimate strength	2.4e+8 Pa	3.131e+08 Pa
Yield strength	1.5e+7 Pa	2.592e+08 Pa
elasticity	69000000 Pa	69000000 Pa

Table 1: Comparison of material properties within the two programmes

For the mesh simulation a beam fixed at one end with a uniform load across its surface was simulated and only the mesh size was altered between the 2 simulations, the mesh simulation was only carried out in Solidworks. The Mesh size used for the simulations can be found in table 2 below. The fine Mesh size had a uniform size throughout, the coarse mesh had a range of sizes. All the Mesh sizes were automatically generated by the computer programme.

Fine mesh size		Coarse mesh size	
Uniform size	7.68223mm	Max	30.7289mm
		Min	15.3645mm

Table 2: Mesh sizes between 2 Mesh validation simulations

Validation results

The results from the two simulation programmes and the equations can be found in this section, broken down into four subtitles to cover the different validation simulations. There will be some simulation result screen shots presented, with all data and screenshots being available in Appendix B.

Single fixed uniform load

Simulation	Solidworks	Ansys	Equation
Max Deformation	2.281mm	2.2818mm	2.2818mm

Table 3: Cantilever beam universal load results.

Single fixed-point load

Simulation	Solidworks	Ansys	Equation
Max Deformation	5.996mm	6.0866mm	6.0849mm

Table 4: Cantilever beam point load results

Double fixed uniform load

Simulation	Solidworks	Ansys	Equation
Max Deformation	0.05097mm	0.050069mm	0.0475mm

Table 5: Supported beam universal load results

Mesh size

Simulation	Fine	Coarse
Max Deformation	2.284mm	2.277mm

Table 6: Cantilever beam universal load mesh size results.

Conclusion

The results shown across all four simulations are comparable. The two simulation programmes and the mathematical calculation all gave results within a small range showing the simulations are accurate and representative of real-world environment.

Due to the specific dimensions chosen to be used as the validation exercise (chosen to better represent the stand) it was not possible to find an existing external case study to compare the results. Further analysis would be recommended if the design of the equipment stand was to progress from a theoretical stage to a physical item. This further analysis should aim to find and identify an existing case study using an Aluminium T6 6061 beam, with dimensions used in this case study replicated in the same methods in the four simulations and mathematical calculations described within this report.

The other validation method adjusting mesh size showed that the changing the mesh size for this simulation had a minimal impact on the overall results of the simulations. Proving the mesh size is not a contributing factor to the overall success of the simulation for this type of model. If the design stand was of a more complex or smaller design then adjusting the mesh size accordingly should be done as due course.

With all the above in mind it has been deemed the simulations of Solidworks and Ansys all share a similar operating process and thus can be deemed representative of a real-world environment. Providing confidence, the stand being simulated and tested, will react in a predictable and representative manner. The recommendation of further validation efforts in the paragraphs above will only add to this confidence level.

Design simulation

The initial design as depicted in figure 1 above was put through a series of simulations on Solidworks and Ansys to provide a comparison between the two CAE systems. Due to the validation process above showing the two programmes are comparable it was deemed appropriate to use both programmes to show the design is responding similarly between the two and thus providing confidence the design is robust and will be able to support the weight its designed to hold.

The two simulations also included two different weight parameters within the simulations. The rationale behind using two different weights for the simulation were due to the uncertainty (and classified data) around the real-world weight distribution from equipment planned to be mounted on top of the stand. The weight of the equipment is stated at 200Kg (2000N) for this report and to represent the two weight differences used in the simulations, a universal 2000N placed onto the three contact points (as shown in figure 2 above). The other load was a total of 2000N spread over three contact points, 786.8N were placed on the two T-bar contact points and 521.5N was placed on the crossbar contact point. The calculations for load distribution can be found in Appendix A. By using these two approaches it can be shown the design can withstand a representative load on top and also the worst-case scenario of 2000N being placed on each individual contact point, thus providing confidence a real-world version of the final design can achieve the job it was designed to do.

The results of the simulations were split into three results, total deformation, Stress (Von mises) and strain (equivalent). This was done to show the areas of the design suffering the most during the simulations, allowing for design improvements to be better incorporated into the final design. The bulk of the screen shots obtained during these simulations showing the various orientations of the model can be found in Appendix C. Key screen shots will be shown in the next section below, showing a comparison between the two systems for reach results output.

During the simulation best efforts were made to keep the mesh sizing consistent across all simulations in both programmes. This was done to help promote similarities between the results from both programmes thus providing better read across between the results. The design was also fixed in place during the simulations in such a way so as to represent the real-world fixing points when the stand is in use. By doing so it would represent a real-world load path for the weight being transferred from the three contact points to the ammunition trolley when in use. Figure 4 below shows the points at which the stand was fixed for all the simulations.

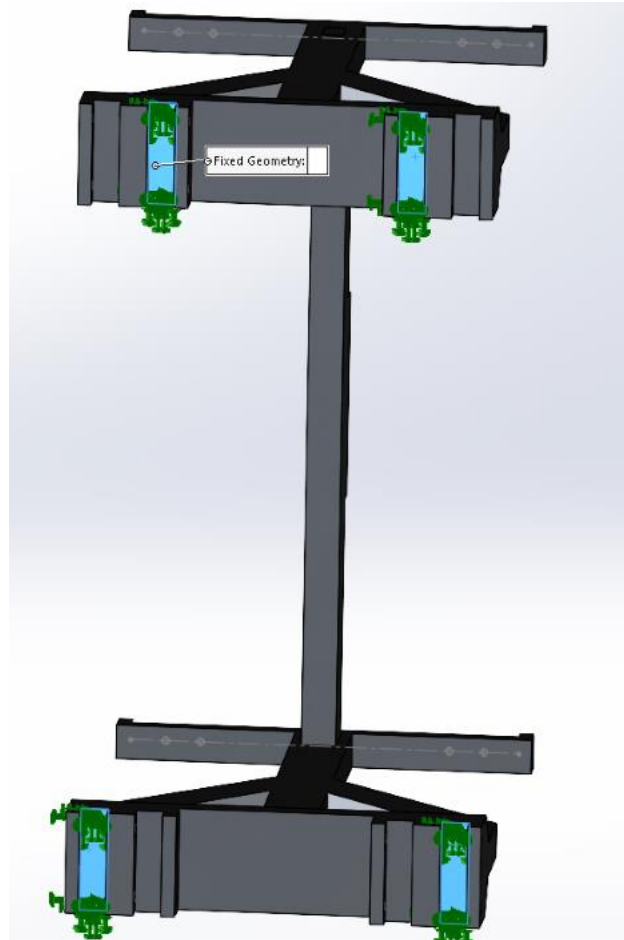


Figure 4: Fixing points for the simulations

Simulation results.

In this section a snapshot of the simulation results for version one of the designs have will be shown. All the results screenshots will be shown in Appendix C.

Figures 5 and 6 below show the maximum amount of deformation seen with 2000N placed on each contact points. With table 7 showing a comparison between the maximum deformation shown by each simulation programme. Solidworks predicts a greater deformation than Ansys, this could be attributed to Ansys being a focused simulation programme where as Solidworks is primarily a CAD programme. It can be assumed Ansys will have a greater degree of accuracy when it comes to more complex modelling unlike the validation exercise.

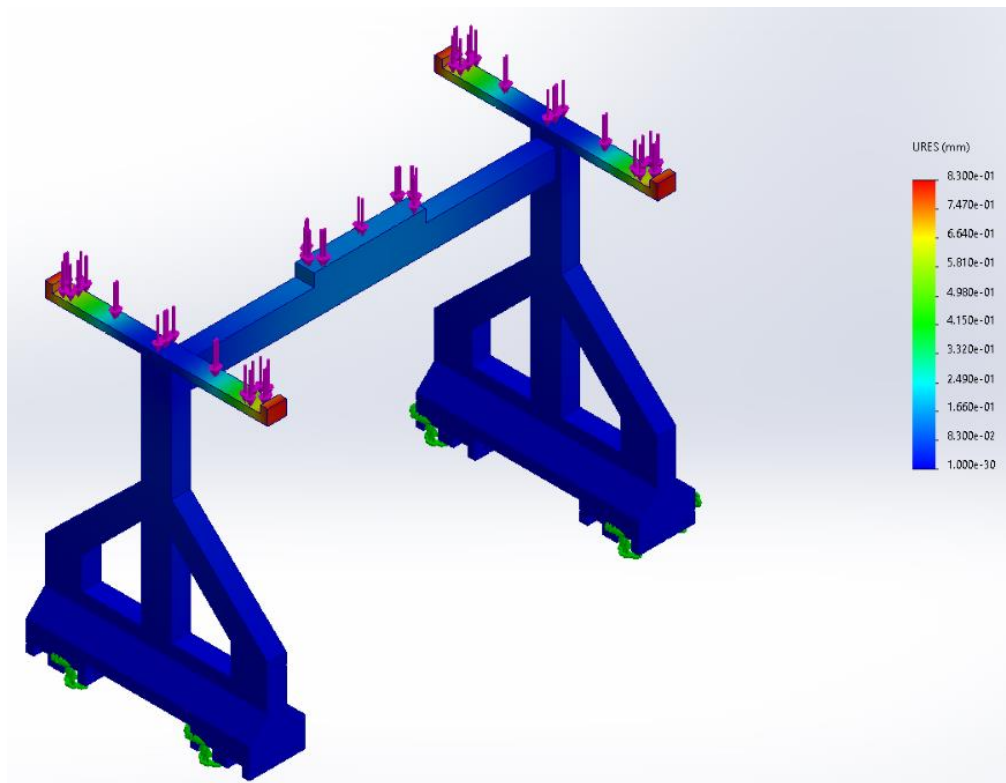


Figure 5: Solidworks simulation with 2000Ns showing deformation.

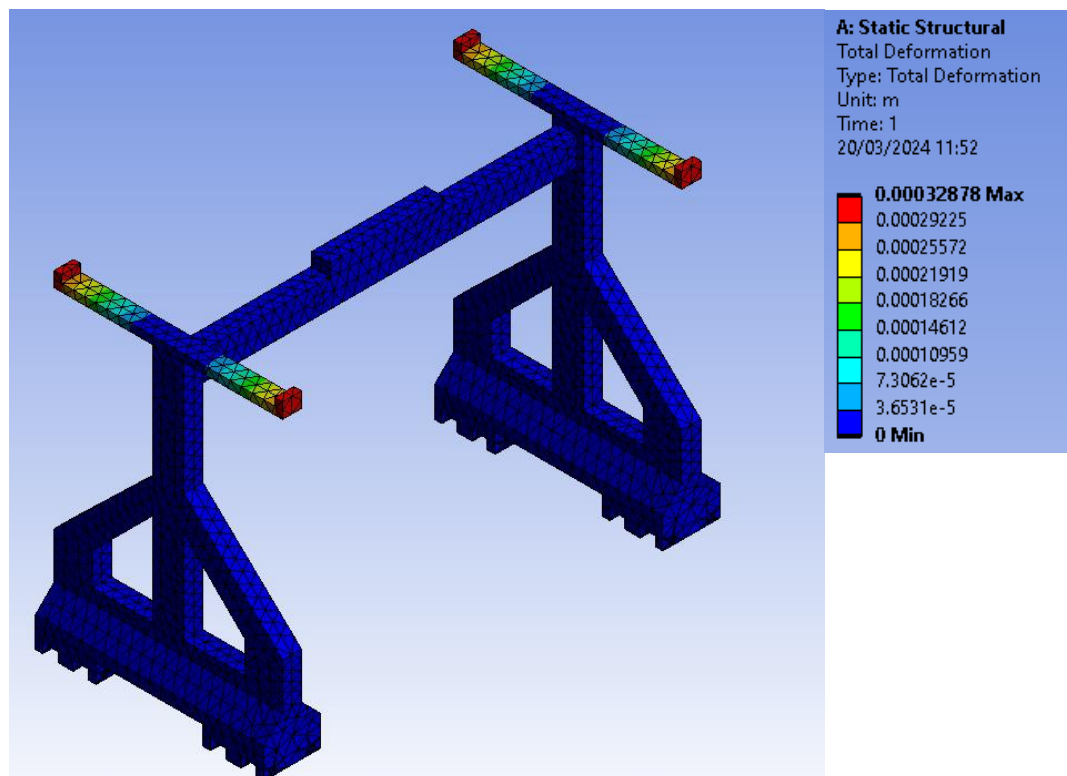


Figure 6: Ansys simulation with 2000Ns showing deformation.

Result parameter	Solidwork	Ansys
Max Deformation	0.83mm	0.32mm

Table 7: Simulation 1 results

Figures 7 and 8 shown total deformation when the 2000N is evenly spread out over the three contact points. Solidworks is again showing a great deformation in the simulation results over Ansys, but both programmes show a reduced total deformation amount.

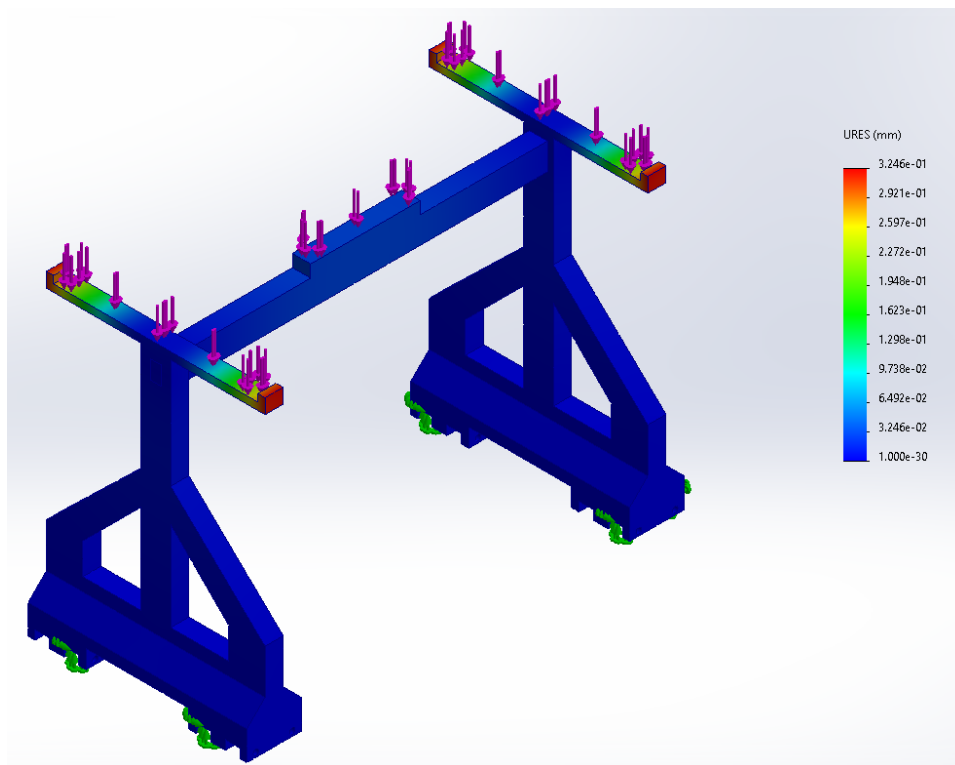


Figure 7: Solidworks simulation with 2000Ns evenly spread showing deformation.

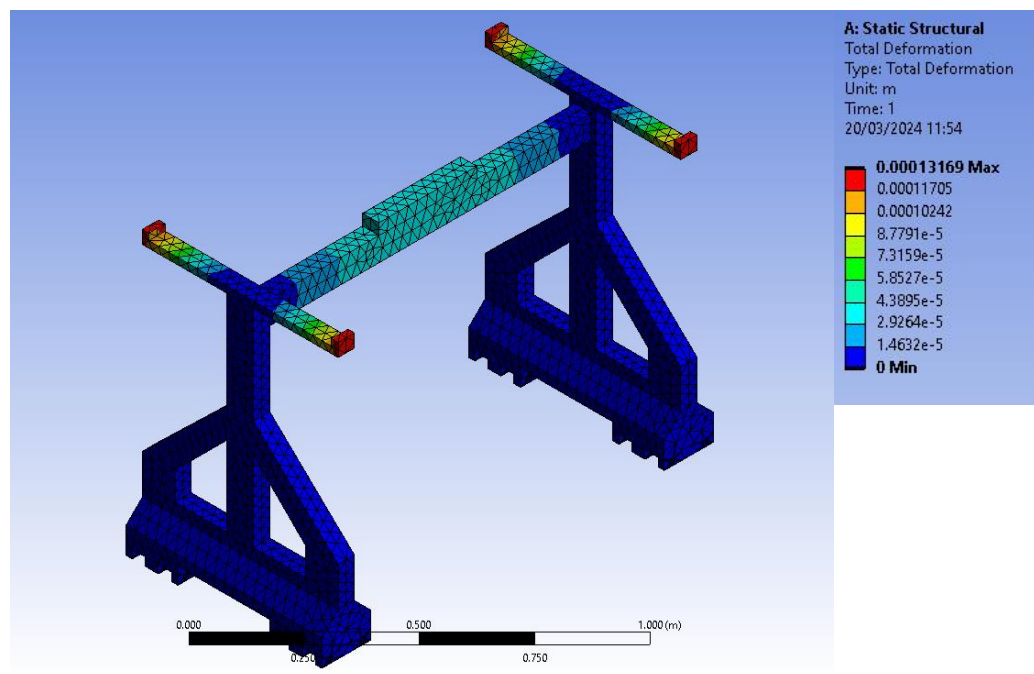


Figure 8: Ansys simulation with 2000Ns evenly spread showing deformation.

Result parameter	Solidwork	Ansys
Max Deformation	0.3246mm	0.13169mm

Table 8:Simulation 2 results

For simplicity and to prevent confusing the next series of screen shots contained within this section are representative of the worst-case scenario of 2000N being placed on all three contact points. The same results for the evenly spread load can be found in Appendix D. The reason for this approach in the report is to show case the main points of stress within this design and to highlight where design changes should be made to help mitigate these stress points.

Figures 9 and 10 below show the stress locations seen within the design when 2000N is placed on each of the contact points.

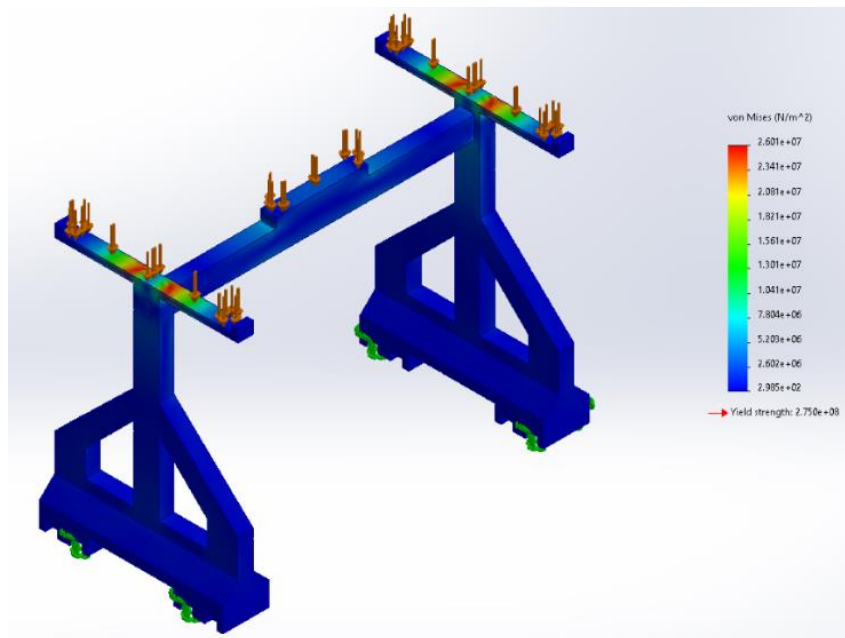


Figure 9:Solidworks simulation with 2000Ns showing Stress.

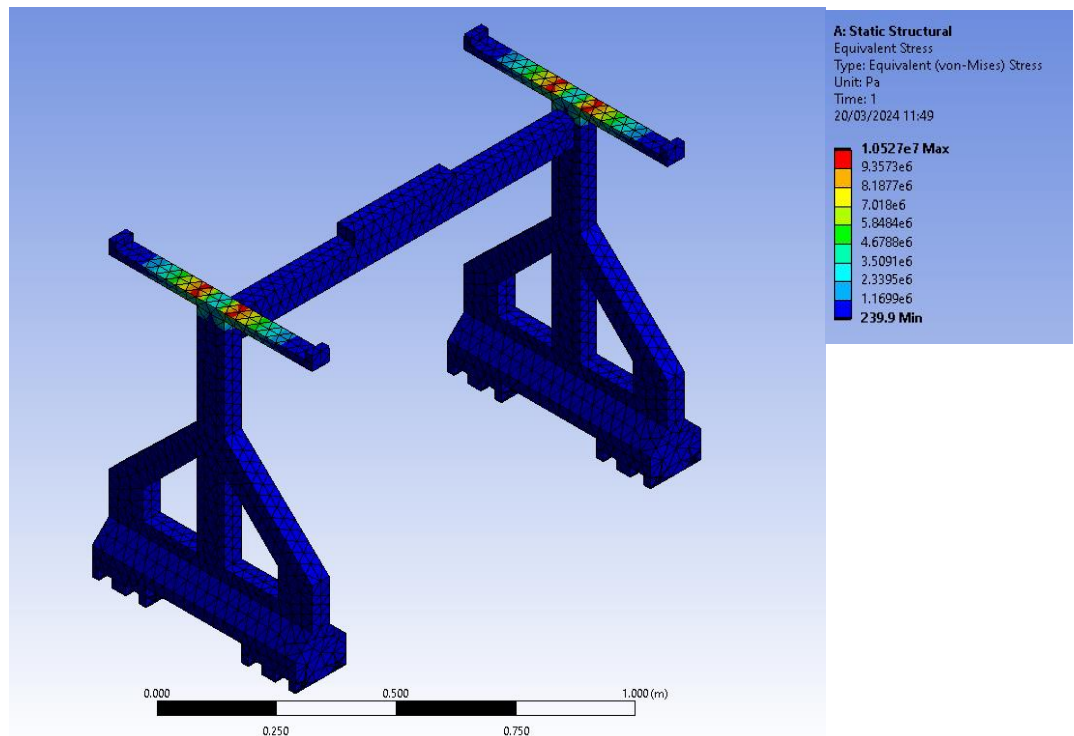


Figure 10: Ansys simulation with 2000Ns showing Stress.

Figures 11 and 12 shown the strain experienced by the design when 2000N are placed on each of the contact points. The results are very similar between the two programmes.

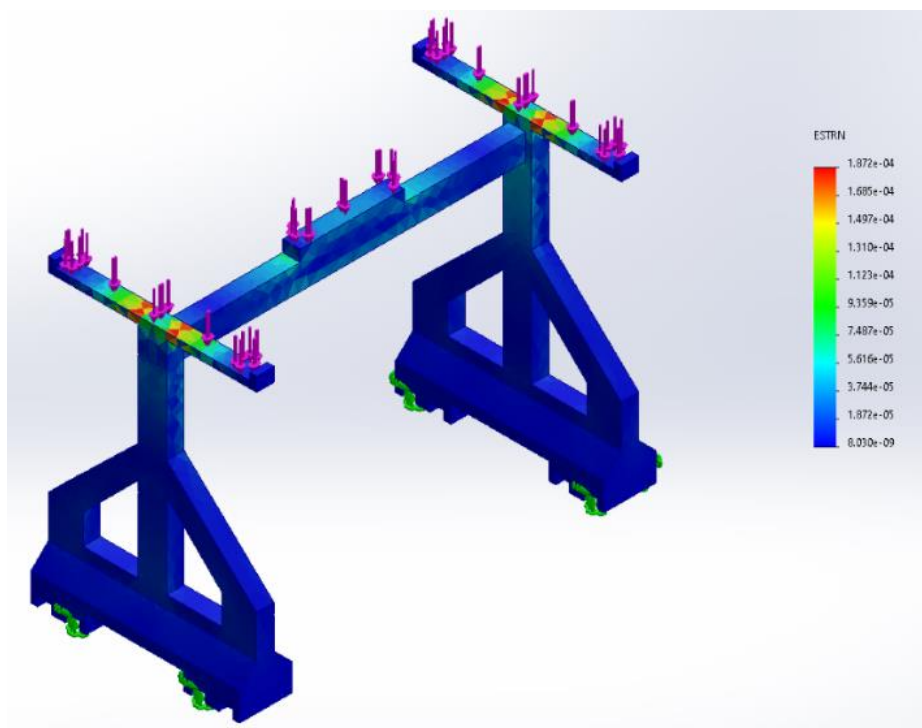


Figure 11: Solidworks simulation with 2000Ns showing Strain.

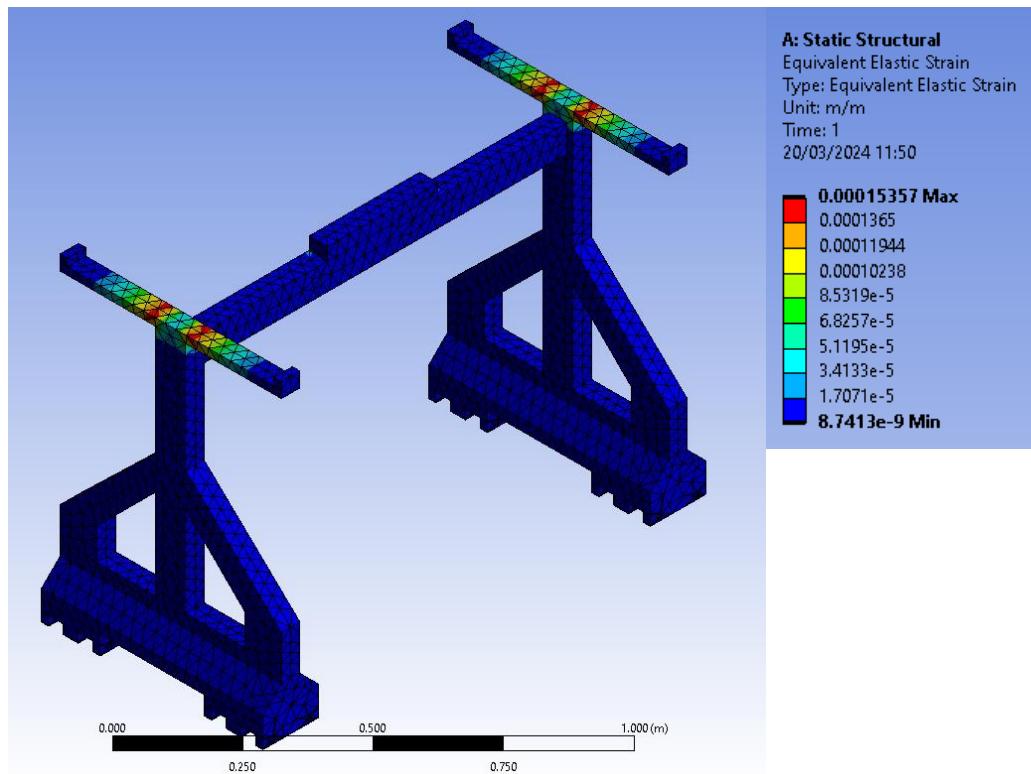


Figure 12: Ansys simulation with 2000Ns showing Strain.

The main area of concern within design 1 is the excessive deformation seen on the extreme ends of the t mount and also the areas of high stress and strain seen where the arms of the T-bar meet the vertical part of the end mounts. These areas have helped guide design improvements to mitigate these problem areas.

Design improvements.

Following the first set of simulations a design change was implemented to help alleviate the main stress locations and to reduce the total amount of deformation with the loads applied across its contact points.

Figures 13 and 14 below show the addition of diagonal support arms placed between the vertical section of the end mounts and the arms of the “T” section of the end mounts.

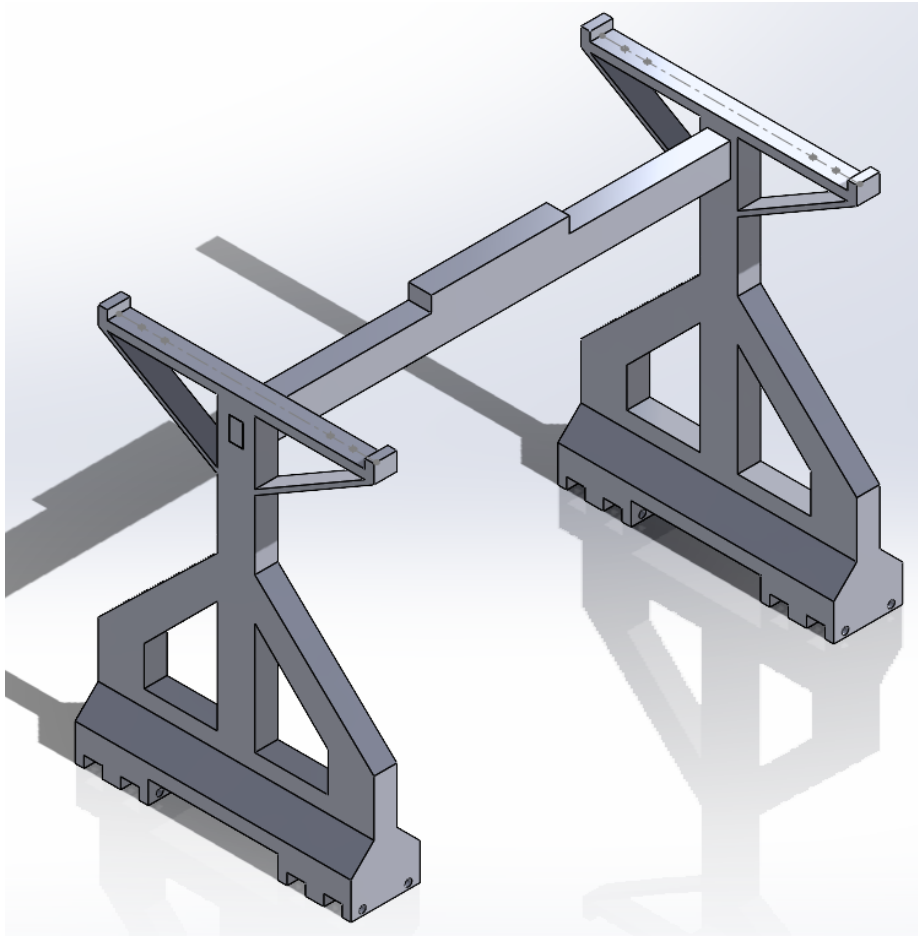


Figure 13: Design version 2

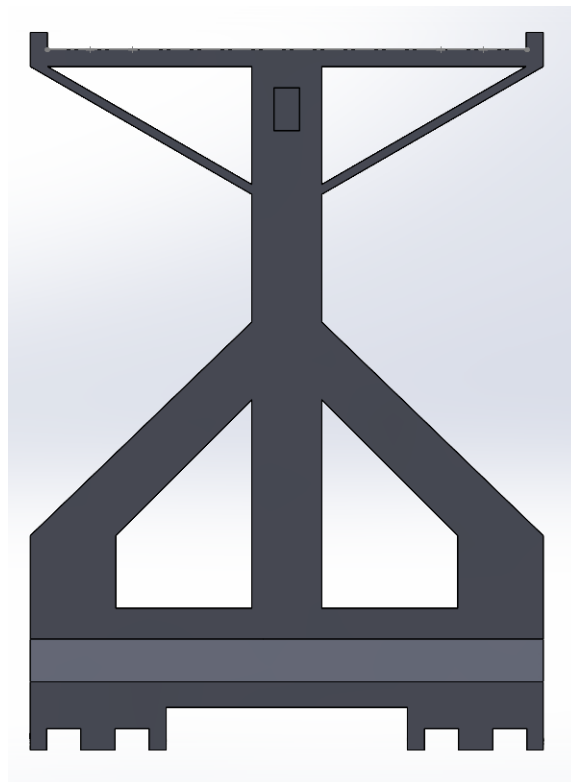


Figure 14: Design version 2 front view

Following these design changes version 2 of the design was subjected to the same series of simulations as version 1 of the design. All of the screen shots from version 2 of the design can be found within Appendix C.

Figure 15 below shows max deformation when 2000N's are placed on each of the contact points, the total deformation shown down is 0.03mm showing a huge reduction compared to version 1 of the design. Its worth noting the results show the centre of the model- coloured red, which typically can be interpreted as showing a “bad” result. This is not the case for this simulation, the red area is only there to highlight the maximum deformation for the whole simulation. If you compare the red sections between version 1 and version 2 designs, the version 2 design red section has a reduction in deformation of 0.29mm. Going from 0.32mm of deformation to 0.03mm, this is a large reduction in total deformation, to the point of no movement at all.

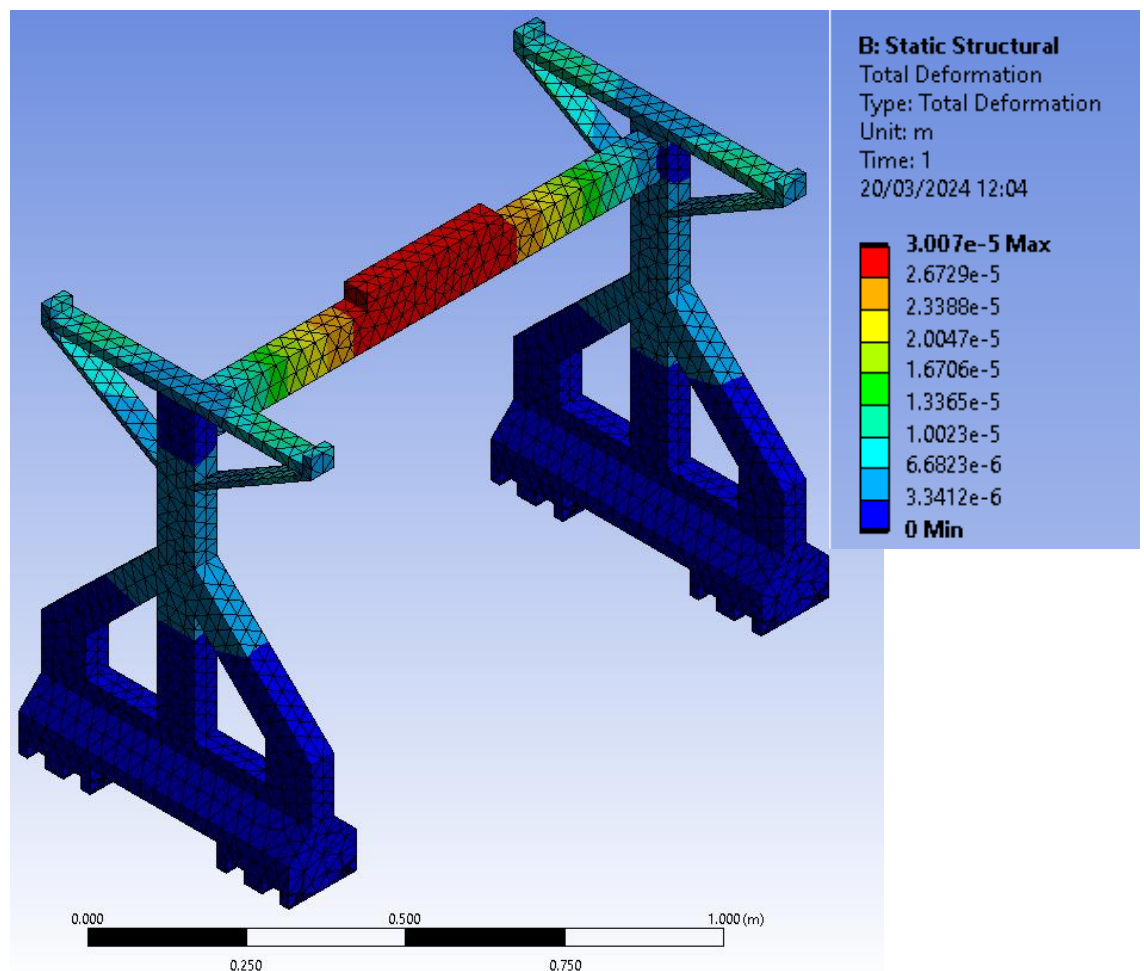


Figure 15: Design version 2 200N simulation showing deformation.

Figure 16 below shows the stress experienced within version 2 of the design, compared to version 1 of the design (figure 10 above) the reduction in stress experience by version 2 is a significant reduction. From to maximum stress experience of $1.437\text{e}6$ Pa compared to version 1 which experienced $1.0527\text{e}7$ Pa of stress. It can also be seen that the area this maximum stress is experience in is greatly reduced.

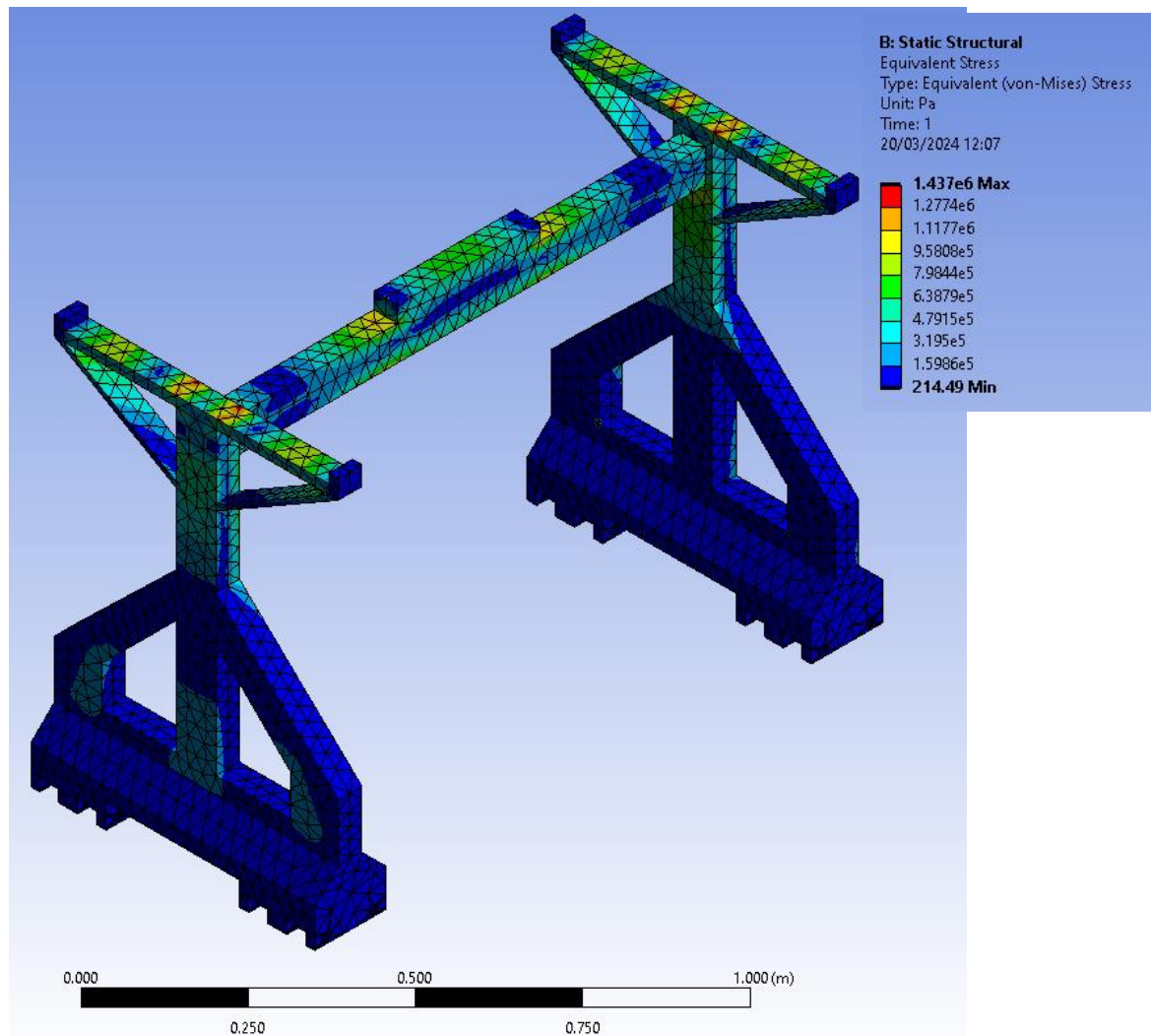


Figure 16: Design version 2 2000N simulation showing Stress.

Figure 17 below shows the strain experienced by version 2 of the design, As with the previous simulation results this also shows a significant reduction in strain. Going from 0.15357 mm/mm of strain to 0.02959 mm/mm of strain. It can also be seen that the strain has been spread evenly over the whole design and not concentrated in one section like before.

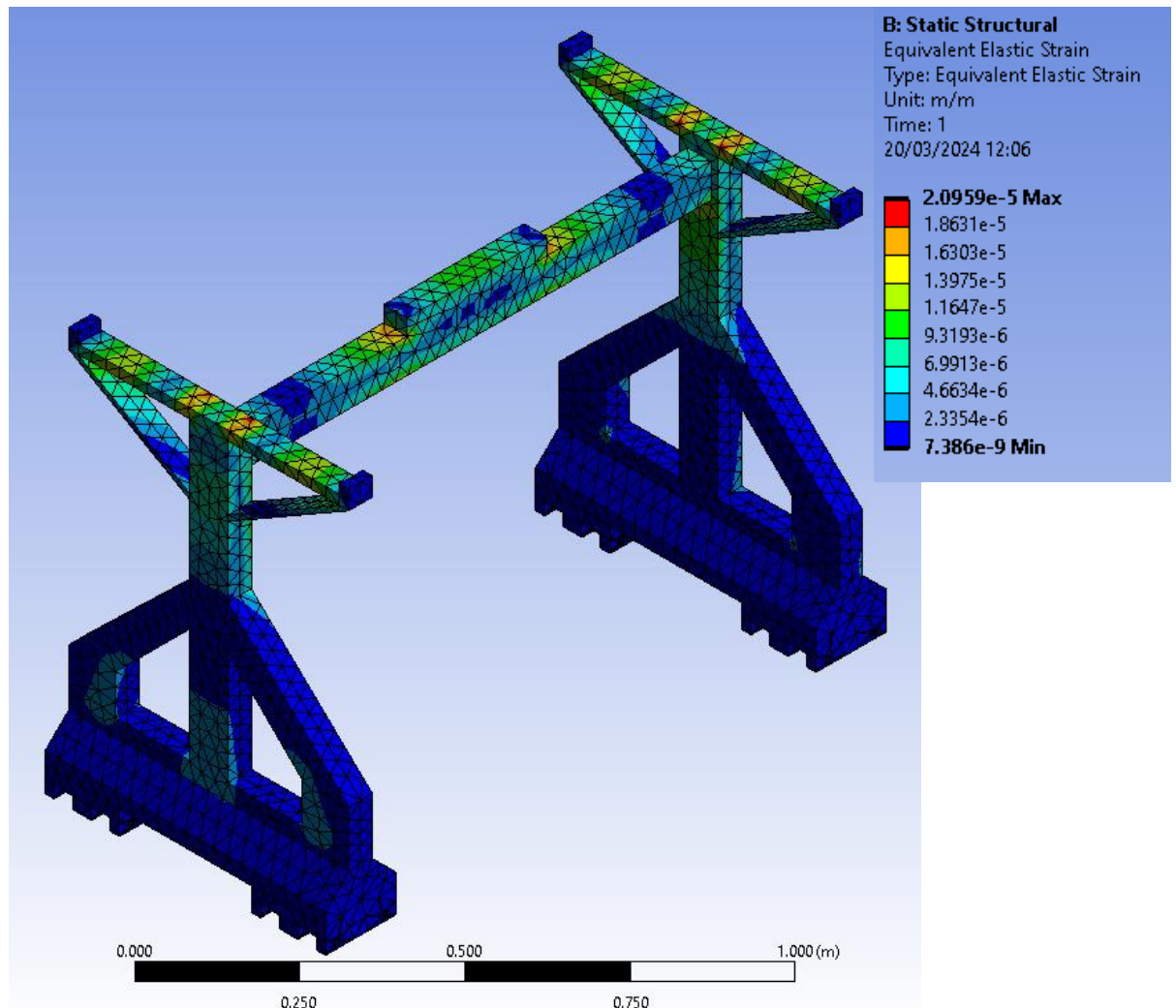


Figure 17: Design version 2 2000N simulation showing Strain.

Design evaluation

For the design evaluation each requirement stated in the requirement section will be briefly discussed and explaining why it is deemed to have met the design requirement.

1. The stand must be able to be mounted to an MHU-191 ammunition trolley.

As can be seen in figure 4 above the shape of the bottom of the stand has been designed to fit onto the MHU-191 ammunition trolley. As can be seen below in figure 18 there are two different widths of rails to secure the stand to. The design of the stand has been done in such away that either end of the stand can fit on both the wide and narrow bars securely, reducing the need of similar parts with minor changes. The design implemented uses a common design that incorporates both the wider and narrower rail attachments into the base of it.

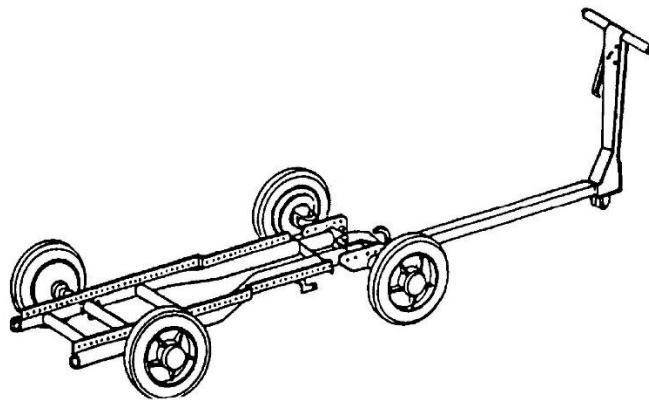


Figure 18: MHU-191 ammunition trolley

2. The stand must be able to accommodate the dimensions of the equipment in question.

Although this requirement cannot be verified the same as other requirements on this list due to the classified nature of the equipment in question. But it can be argued that the dimensions although not representative of the real-world equipment can be adjusted to meet this requirement if this stand was to progress from a theoretical project, applying the appropriate security markings to the design output will allow for the correct dimensions to be used.

3. The stand must be able to hold a weight of 200kg's with no more than 1.5mm of displacement in its structure.

As has been demonstrated with the simulations above design version 2 can hold the 200kg/2000N weight on top of the 3 contact points without deforming more than 0.03mm. Thus, proving this requirement has not only been met but exceeded as well.

4. The stand must spread the weight over an area 5000 cm^2 .

As with requirement 2 the dimensions shown in this report are not representative of the real-world equipment. But with that said the calculations contained within appendix A the three contact points spread the 200kg/2000N over a surface area of 7100cm^2 exceeding the requirement of 5000cm^2 . As with requirement 2 the design can be adjusted to meet the real-world equipment if this project proceeds to a real-world example.

5. The stand must keep the equipment secured upon itself with no additional lashings.

With the inclusion of the vertical stays placed at the end of each of the T-arms, the overall design can be argued to meet this requirement as they prevent the equipment sliding off the ends of the arms. As with requirement 2 and 4 the dimensions can be adjusted to better match the real-world equipment if required.

6. The stand must allow for ease of access to the top of the equipment at a height of no less than 1 meter above the ground.

Although this requirement cannot be verified the same as other requirements on this list due to the classified nature of the equipment in question. But it can be argued that the dimensions although not representative of the real-world equipment can be adjusted to meet this requirement if this stand was to progress from a theoretical project, applying the appropriate security markings to the design output will allow for the correct dimensions to be used.

7. The stand must be constructed with the use of the minimum number of parts possible to achieve the other stipulated requirements.

This requirement has been met due to the use of only 3 total parts required to assemble the stand, with two of the parts being identical. It could be possible to complete remove the cross bar from the design if further reductions were needed, but this was not considered due to the need to requirement 4.

8. The stand must be capable of being assembled with basic hand tools.

This requirement has been met as the 3 parts making up the assemble do not require any tools to complete the assembly. The only hand tools required to fully assemble this design are needed to integrate the stand onto the MHU-191 ammunition trolley.

9. The stand must be constructed of Aluminium alloy 6061 T6.

At this stage this requirement has not been met by a physical stand, but it can be argued the requirement has been met due to all the simulations and validation efforts using Aluminium ally 6061 T6 as its material properties.

Conclusion

This report has explained the benefits of CAE within the design process of an item, demonstrated why validating a simulation is a key and integral part of this process. Whilst providing an example of an item's design process, taking the requirements, and producing a first draft design to meet these requirements. Demonstrating a validation process for the simulations and subjecting it to a series of simulations and then applying design changes to improve the simulation results.

CAE is the backbone of modern engineering design providing design engineers with valuable tools. These tools allow for faster, accurate and better designs to develop into a real product. By being able to test these designs in a virtual environment before any physical product is ever made helps reduce cost of producing expensive prototypes. This allows engineers to find flaws in their design and also prove their designs work. However as with any tool it has its limitations, the design and simulation outputs are only ever as good as the information put into them. An engineer must have a sound understanding of the physics involved and the specific characteristics of the CAE system being used to fully use these systems to their full potential. A good design engineer will understand that the simulations take place in a sanitary and perfect world when it's on a computer system and the results from these simulations don't always translate exactly to the real world. For example, an electronics engineer designing a complex circuit board can use a CAE programme to account for every volt in the system he is designing, making sure there is no loss in efficiency of the circuit in question. But if he doesn't consider real world tolerance issues with manufacturing of the components that will be used in his design, then the circuit board would never work as well as it does in the CAE simulation.

Validation is a key part of the CAE design process as discussed in previous chapters; without this step a simulation is just that a simulation. There would be no confidence the simulation results are accurate and represent a real-world scenario. By using real world case studies, evidence and data from experiments and testing with real parts and components an engineer can compare these to a validation simulation on the CAE programme. Doing so can provide confidence the simulation behaves in the correct way. In this report a series of beam bending validation exercises were undertaken using mathematical formulas as the comparison data. Unfortunately for this report it was realised at a too late of a stage to find a real-world case study to use to back up the simulation and mathematical simulations that were used. The author of this report should have researched for a beam bending case study before undertaking any validation simulation or calculations, that way the beam bending being simulated would have been of the same dimensions and properties of a real-world example. Allowing for better comparison between all stages of the validation effort, however this was not the case, but the validation efforts within this report were not futile, the validation exercises used a beam of representative dimensions and material properties to that of the final design, providing confidence the simulation does behave consistently across all simulations.

In conclusion providing the engineer using the CAE is competent and knowledgeable enough to understand the complex nature of simulation outputs and can translate the results to the

real-world then CAE is an invaluable engineering tool. Providing great benefits to the industry they are utilised in when employed in a systematic approach to engineering design processes. This report demonstrated that with the use of a design process for a stand to be used on a particular item of equipment. Taking a design from requirements to a working simulation, demonstrating and describe the CAE process including validation efforts to prove the simulations are viable. Further work was recommended to add confidence to the validation and simulation efforts if the item in question was to progress for a paper-based exercise into a physical item.

References

- [1] Dassault Systemes, "Dassault Systemes," [Online]. Available: <https://www.3ds.com/about>. [Accessed 18 03 2024].
- [2] G. Aggarwal, "Internshala Trainings," [Online]. Available: <https://trainings.internshala.com/blog/what-is-ansys/#:~:text=Ansys%20is%20an%20analyzing%20software,or%20finite%20element%20analysis%20software..> [Accessed 2024 03 18].
- [3] HM Government, "Government Security Classifications Policy," HM Government, London, 2023.
- [4] N. Basonry, "Profile Tree," 13 02 2024. [Online]. Available: <https://profiletree.com/advantages-and-disadvantages-of-cad/>. [Accessed 18 04 2024].
- [5] "Cad Crowd," 24 08 2023. [Online]. Available: <https://www.cadcrowd.com/blog/product-testing-and-validation-the-role-of-cad-simulation-services-for-companies/>. [Accessed 18 04 2024].
- [6] J. Bell, "Interpretation of simulation for model-based," Department of Computer Science, University of Wales, Aberystwyth, Aberystwyth, 2006.
- [7] E. Golightly, "Click up," 31 07 2023. [Online]. Available: <https://clickup.com/blog/project-requirements/#:~:text=Project%20requirements%20are%20the%20specific,what%20they%20want%20to%20achieve..> [Accessed 18 03 2024].
- [8] Alamo Woodwork, "Medium," 26 04 2023. [Online]. Available: <https://medium.com/@alamowoods/what-are-the-strengths-and-weaknesses-in-using-mortise-and-tenon-joinery-in-outdoor-furniture-143ec49fc43a> . [Accessed 11 04 2024].
- [9] "Simexlab," 05 04 2023. [Online]. Available: <https://simexlab.eu/blog/the-importance-of-validation-and-verification-in-simulation-modeling-a-comprehensive-guide.html>. [Accessed 28 04 2024].
- [10] "MBDA Consultants," 11 03 2023. [Online]. Available: <https://mbseconsultants.com/5-advantages-of-using-simulation-for-validation/>. [Accessed 28 04 2024].
- [11] D. K. Pace, "Modeling and Simulation Verification and Validation," *JOHNS HOPKINS APL TECHNICAL DIGEST*, vol. 25, no. 2, pp. 163 -172, 2004.
- [12] S. Solmaz, "Simscale," 09 06 2023. [Online]. Available: <https://www.simscale.com/blog/verification-and-validation-cae/>. [Accessed 28 04 2024].
- [13] K. A. & N. Swanson, "OMNI calculator," 17 04 2024. [Online]. Available: <https://www.omnicalculator.com/construction/beam-deflection>. [Accessed 22 04 2024].

Appendix

Appendix A

Calculations

1. Inertia (I) calculation for Beam deflection.

$$\frac{Width \times Height^3}{12}$$

$$\frac{50 \times 72.5^3}{12} = 1587825.521mm^2$$

2. Fixed beam deflection with a uniform load.

$$\frac{\omega L^4}{8EI} = \delta Max$$

$$\frac{2000 \times 1000^4}{8 \times 69000000 \times 1587825.521} = 2.2818mm$$

3. Fixed beam deflection with a point load.

$$\frac{PL^3}{3EI} = \delta Max$$

$$\frac{2000 \times 1000^3}{3 \times 69000000 \times 1587825.521} = 6.0849mm$$

4. Fixed beam deflection at both ends with a universal load.

$$\frac{\omega L^4}{384EI} = \delta Max$$

$$\frac{2000 \times 1000^4}{384 \times 69000000 \times 1587825.521} = 0.0475mm$$

5. Load bearing Surface area.

$$*Length \times Width = Area$$

$$560 \times 50 = 28000mm^2$$

$$50 \times 300 = 15000mm^2$$

$$(28000 \times 2) + 15000 = 71000\text{mm}^2$$

$$\text{Total load bearing area of } 71000\text{mm}^2 \equiv 7100\text{cm}^2$$

**Length and width dimensions taken from measurements of the solidworks model, they are not representative or comparable to the real-world equipment designed to sit on this stand*

6. Load pressure per mm^2 .

$$200\text{KG} \cong 2000\text{N}^*$$

$$\frac{\text{Force}}{\text{Area}} = \text{Pressure}$$

$$\frac{2000}{71000} = 0.0281\text{N/mm}^2$$

**200kg converted to Newtons is 1961.330N, for simplicity in the calculations and simulations the total Newtons were rounded up to 2000N.*

7. Load distributed over each load bearing surface.

$$\text{Pressure} \times \text{area} = \text{Force}$$

T-mounts area:

$$0.0281 \times 28000 = 786.8\text{N}$$

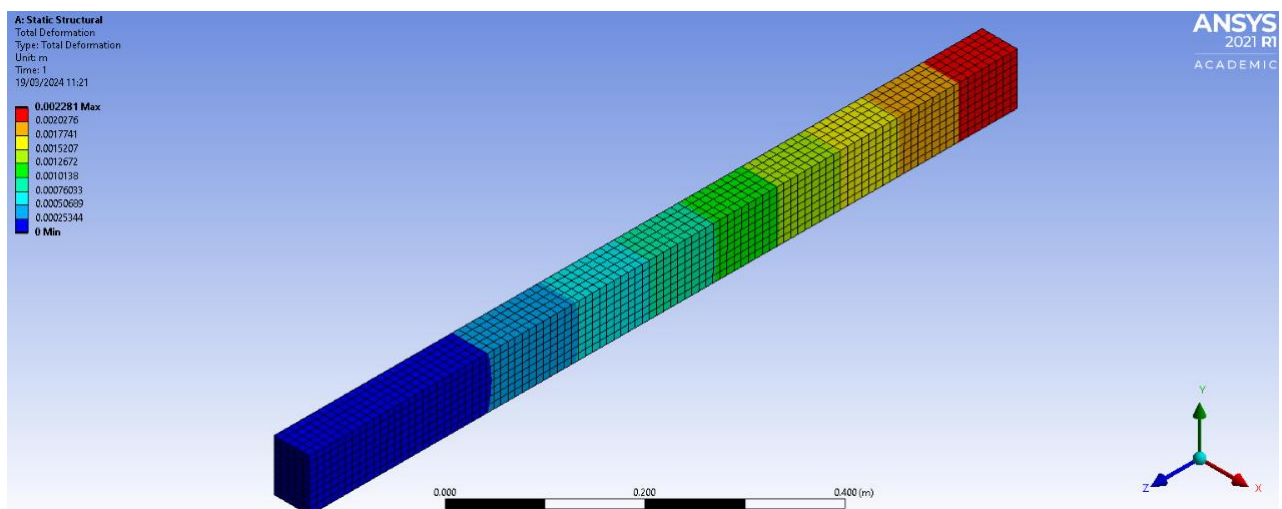
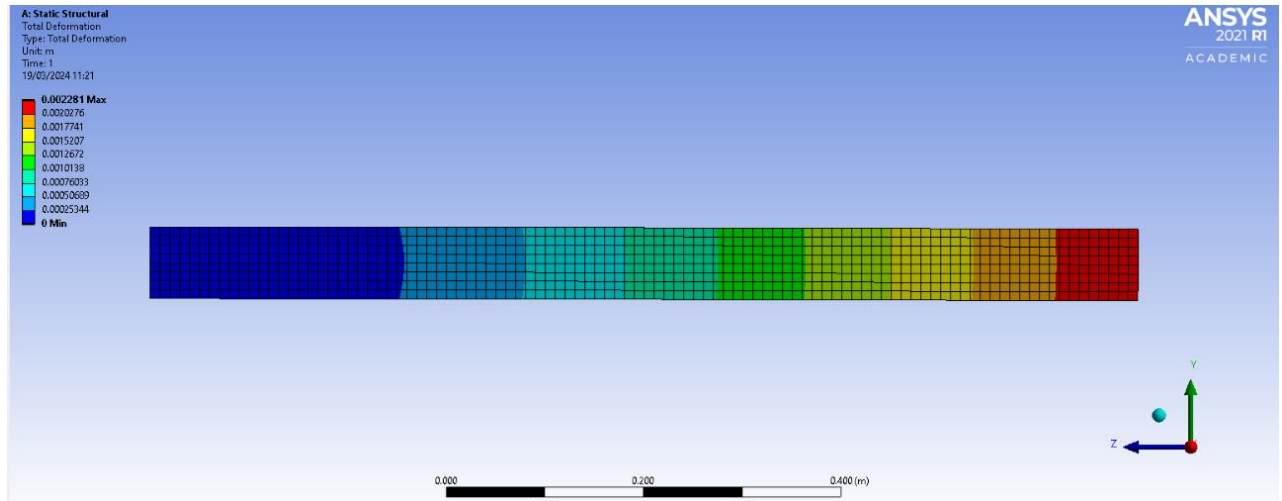
Cross bar area:

$$0.0281 \times 15000 = 521.5\text{N}$$

Appendix B

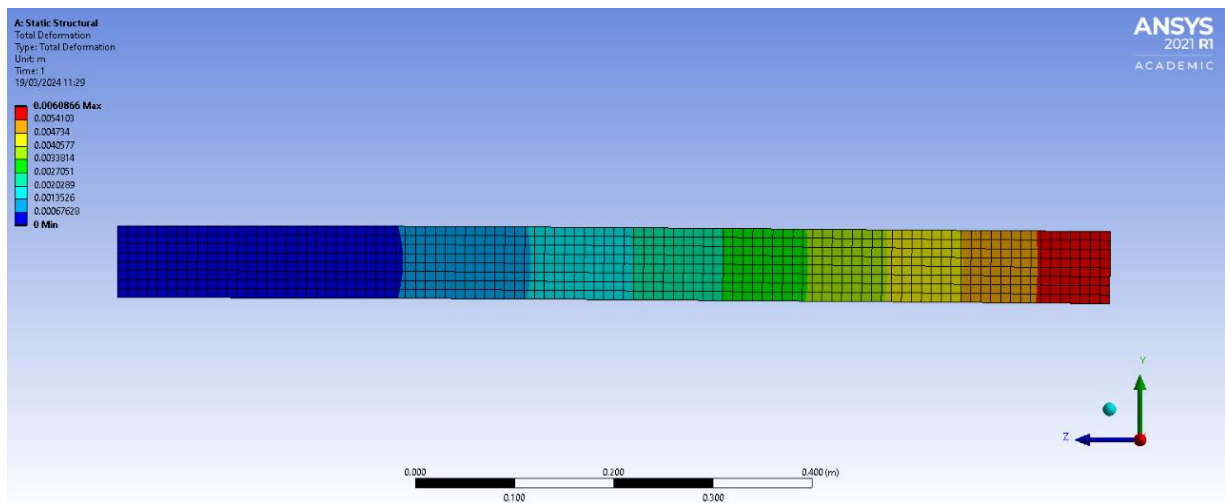
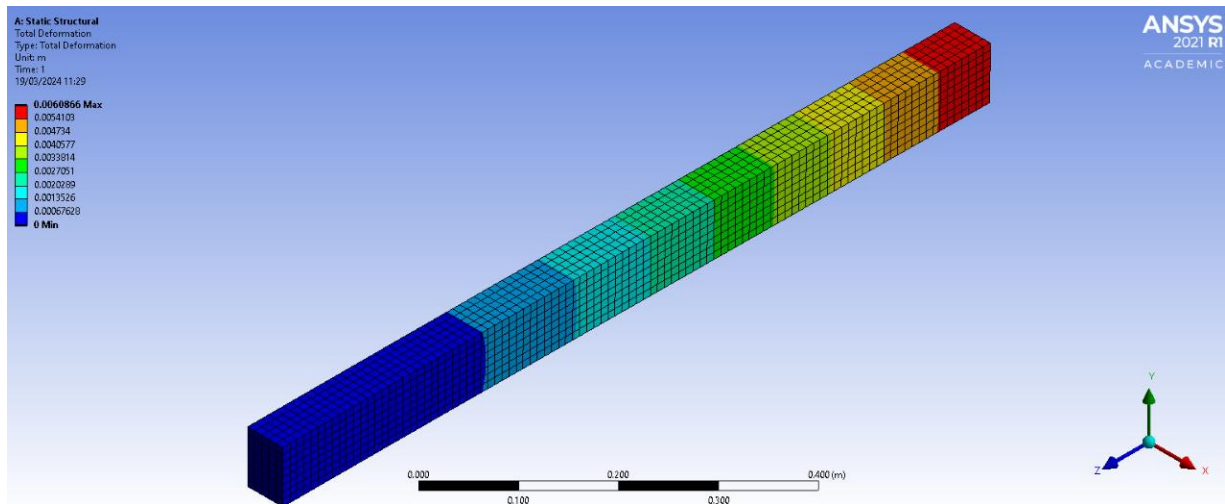
Beam dimensions 1000 (L) x 50 (W) x 72.5 (H) mm Force 2000N aluminium alloy 6061 T6 fixed 1 end in Ansys.

Showing Deformation



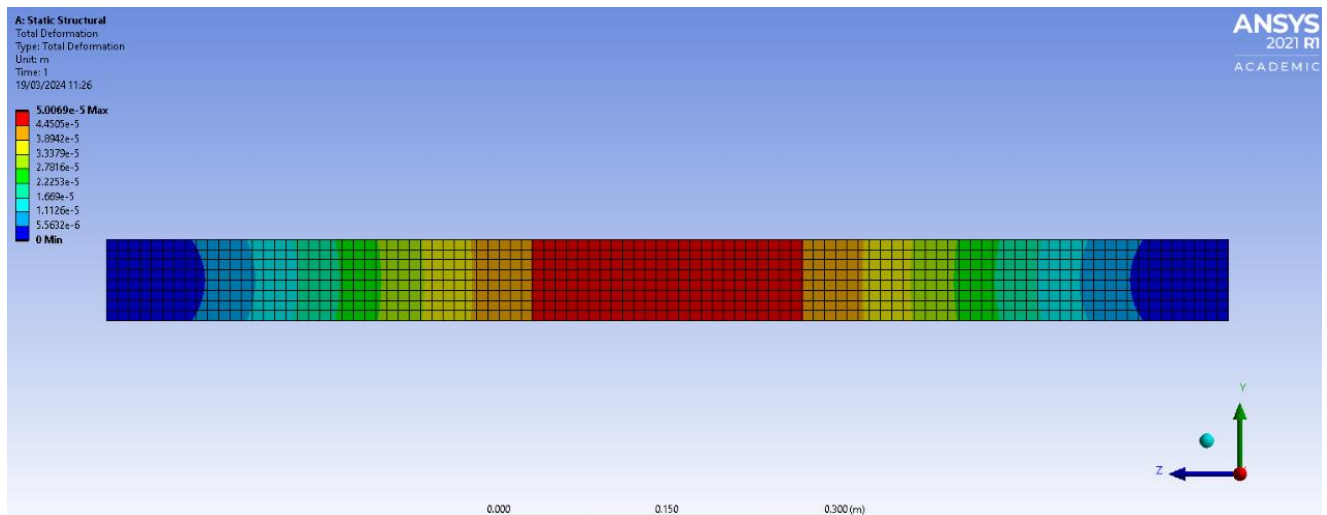
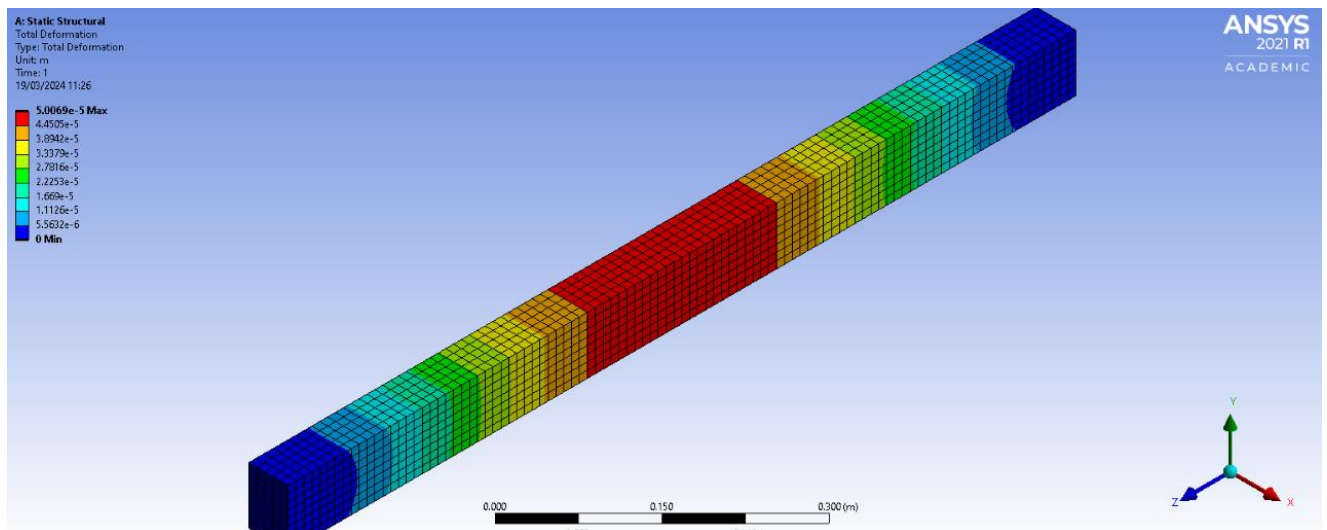
Beam dimensions 1000 (L) x 50 (W) x 72.5 (H) mm Force 2000N aluminium alloy 6061 T6, force on edge in Ansys

Showing Deformation



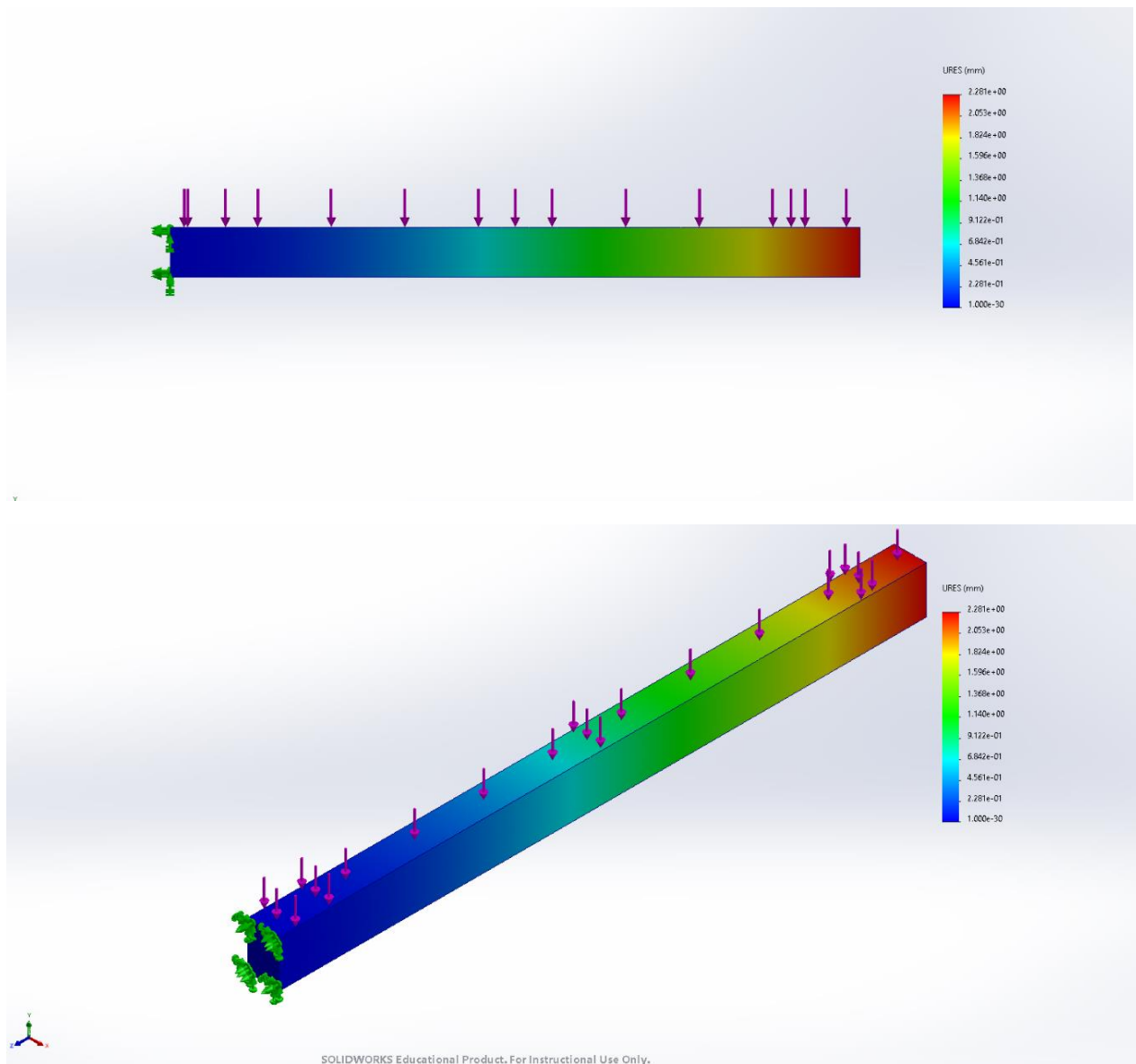
Beam dimensions 1000 (L) x 50 (W) x 72.5 (H) mm Force 2000N aluminium alloy 6061 T6 fixed both ends in Ansys.

Showing Deformation



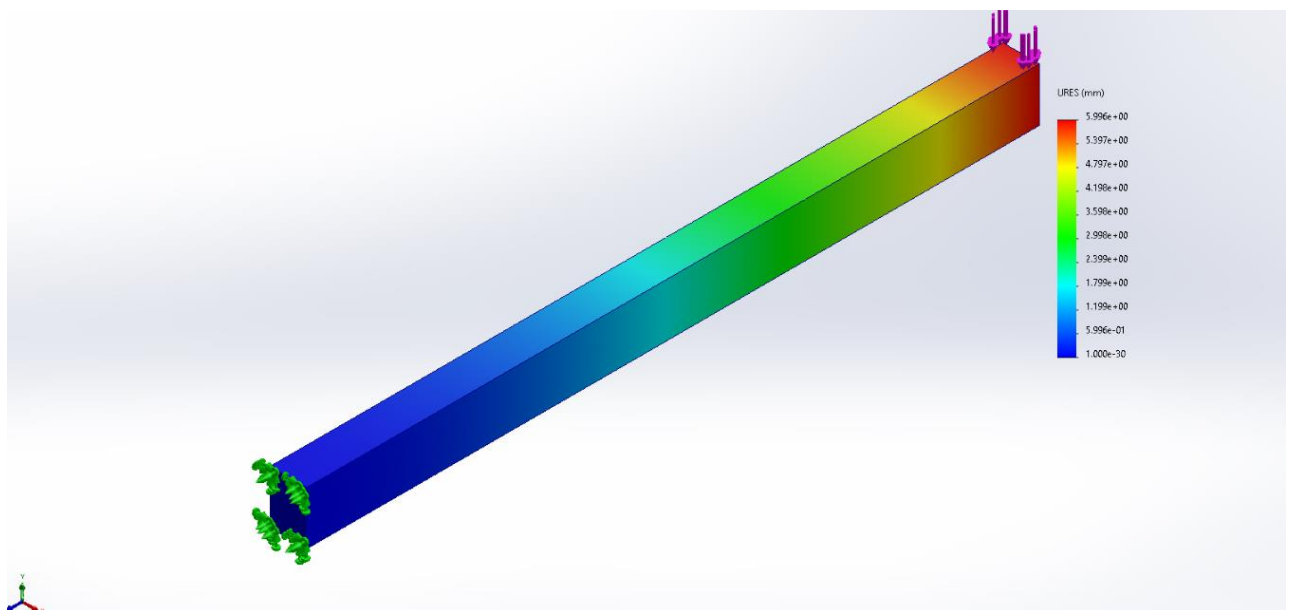
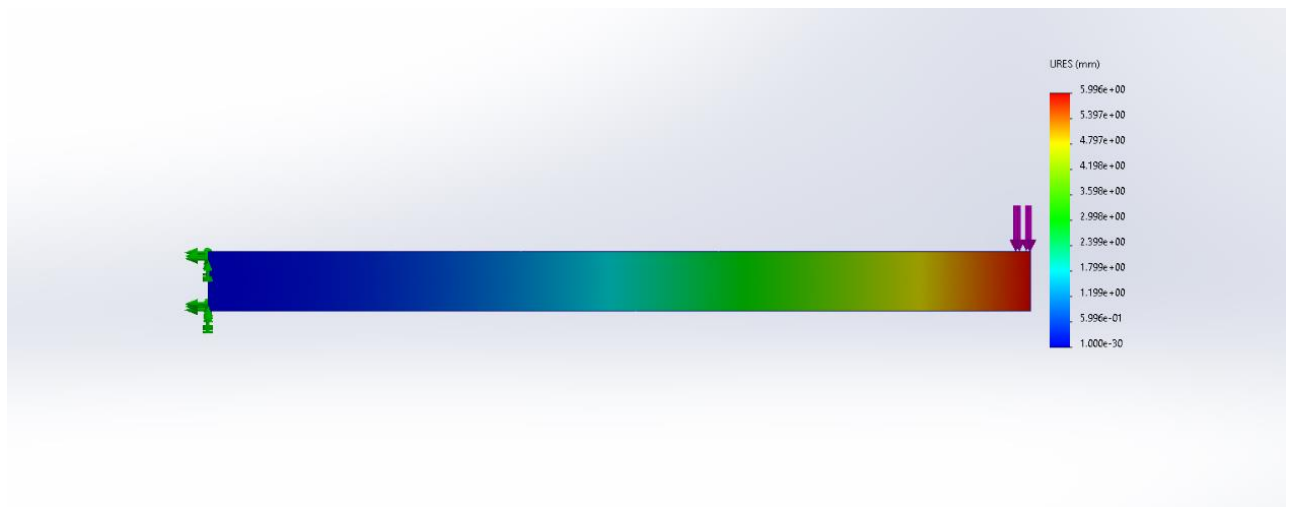
Beam dimensions 1000 (L) x 50 (W) x 72.5 (H) mm Force 2000N aluminium alloy 6061 T6 fixed 1 end in Solidworks.

Showing Deformation



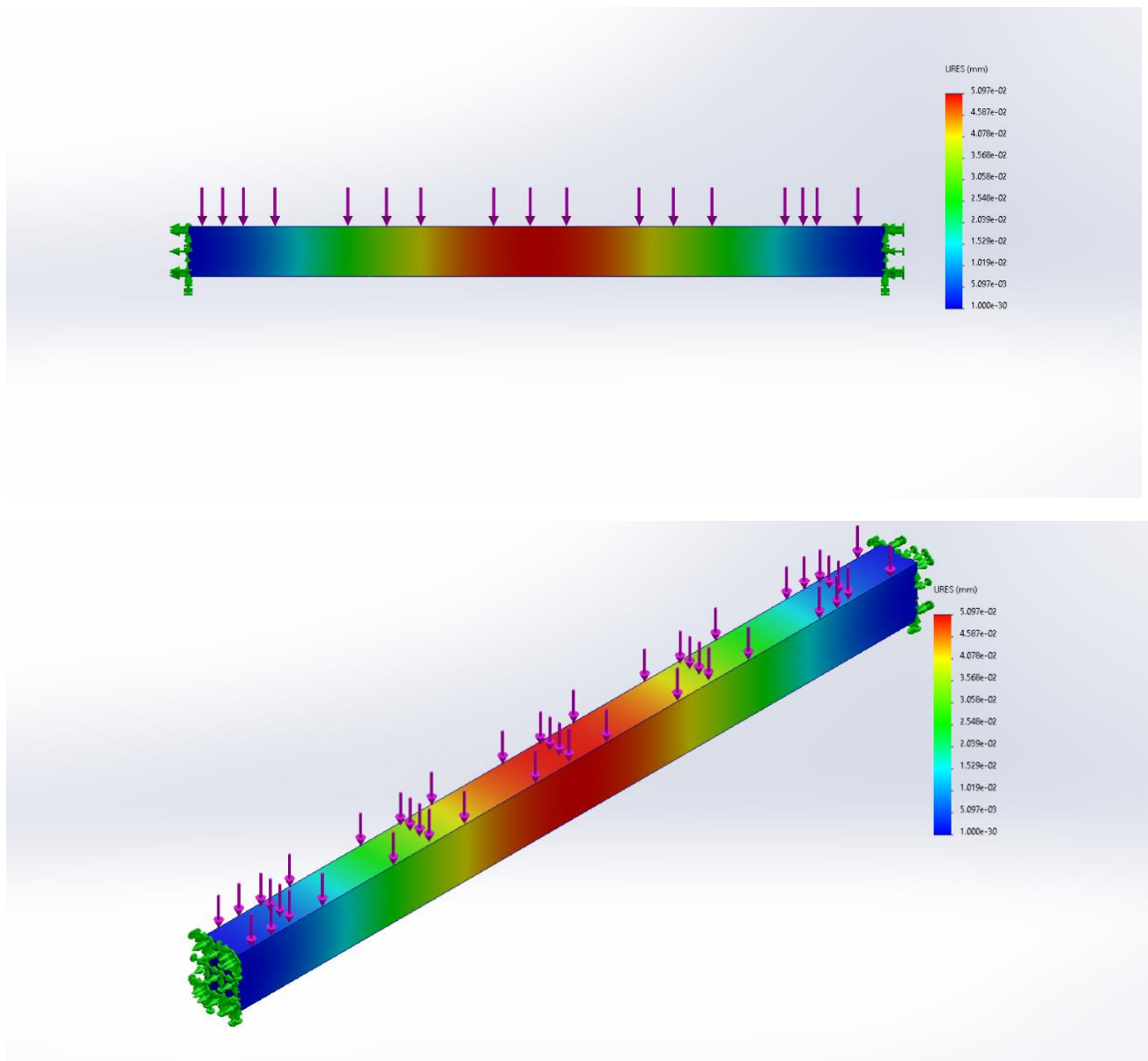
Beam dimensions 1000 (L) x 50 (W) x 72.5 (H) mm Force 2000N aluminium alloy 6061 T6 force on edge in Solidworks

Showing Deformation



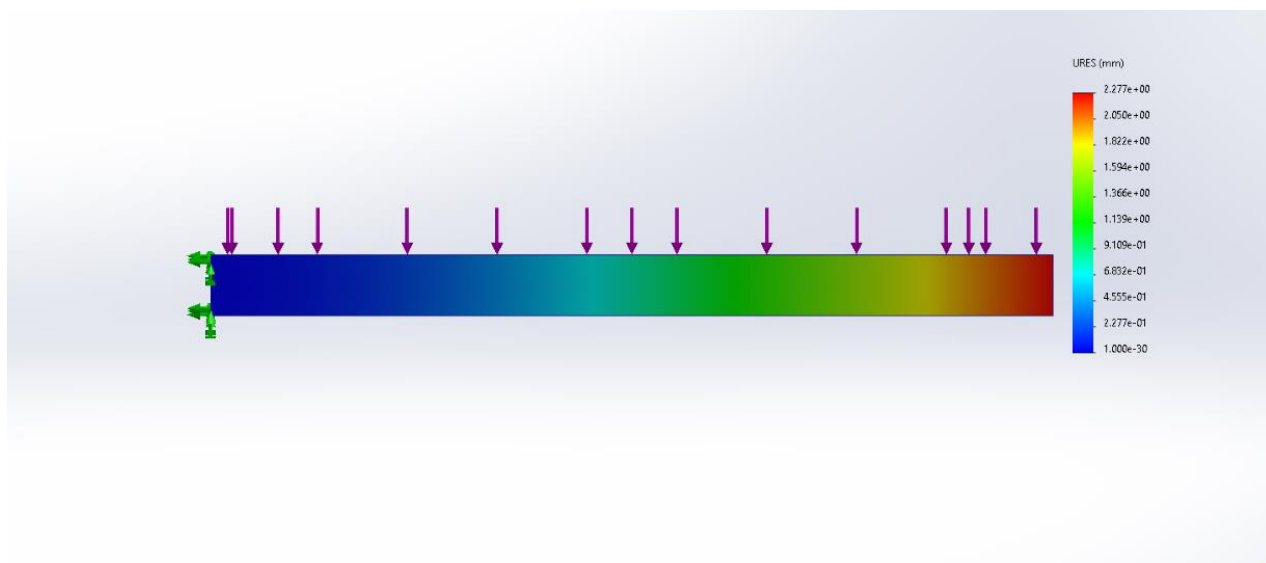
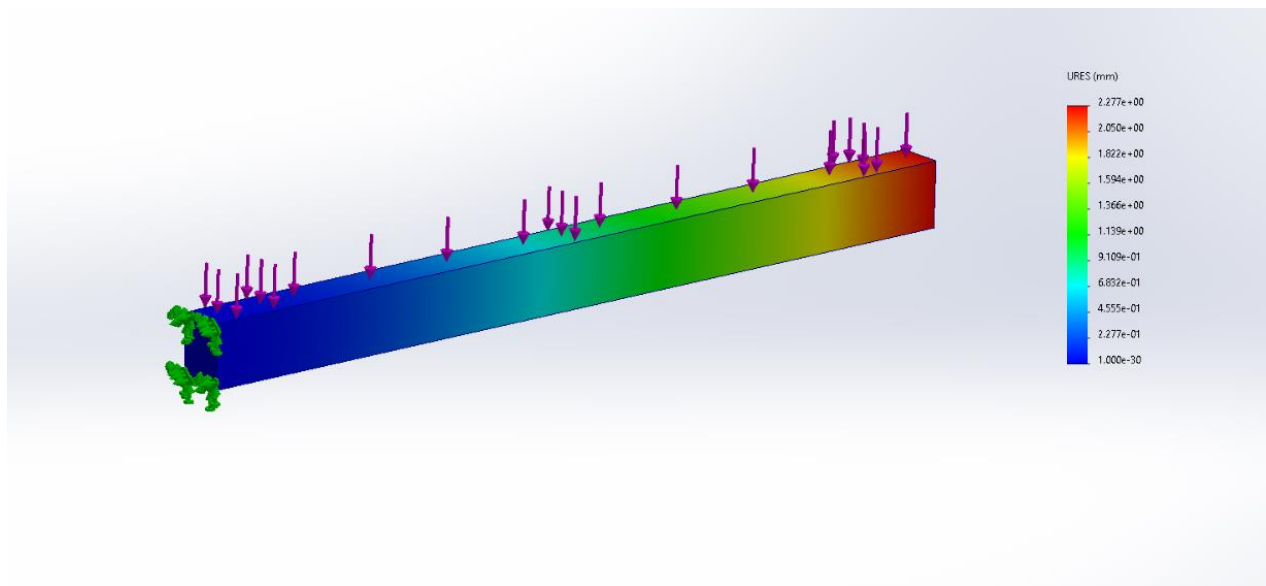
Beam dimensions 1000 (L) x 50 (W) x 72.5 (H) mm Force 2000N aluminium alloy 6061 T6 fixed both ends in Solidworks.

Showing Deformation



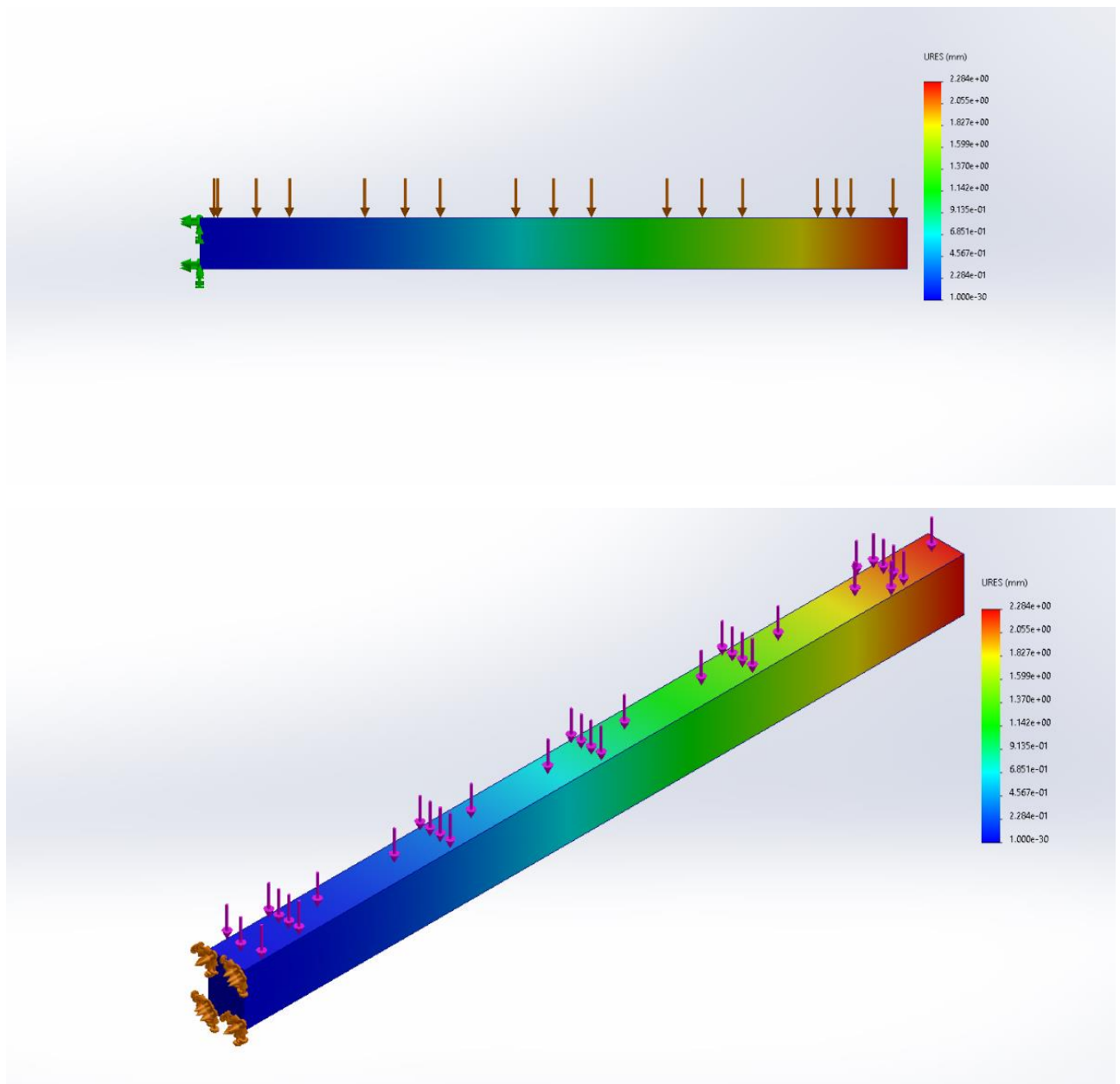
Mesh size Corse (Max 30.7289 mm, Min 15.3645mm) fixed at 1 end uniform load of 2000N across top in Solidworks.

Showing Deformation



Mesh size fine, (Max 7.68223 mm, Min 7.68223mm) fixed at 1 end uniform load of 2000N across top in Solidworks.

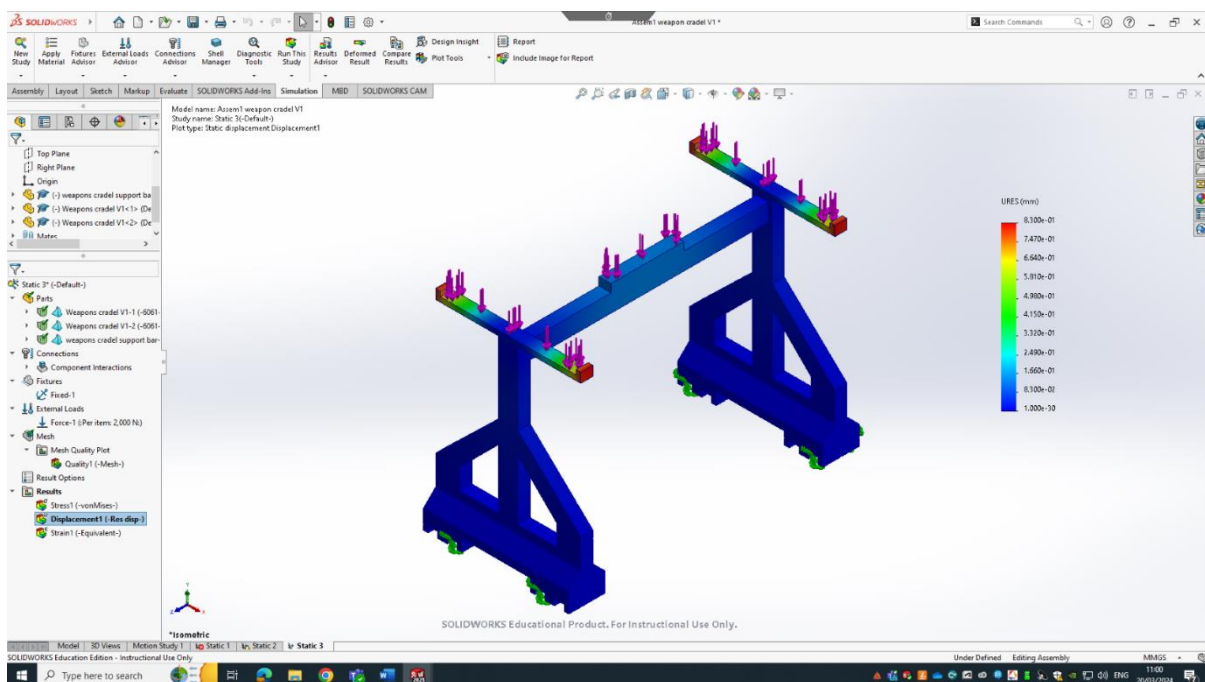
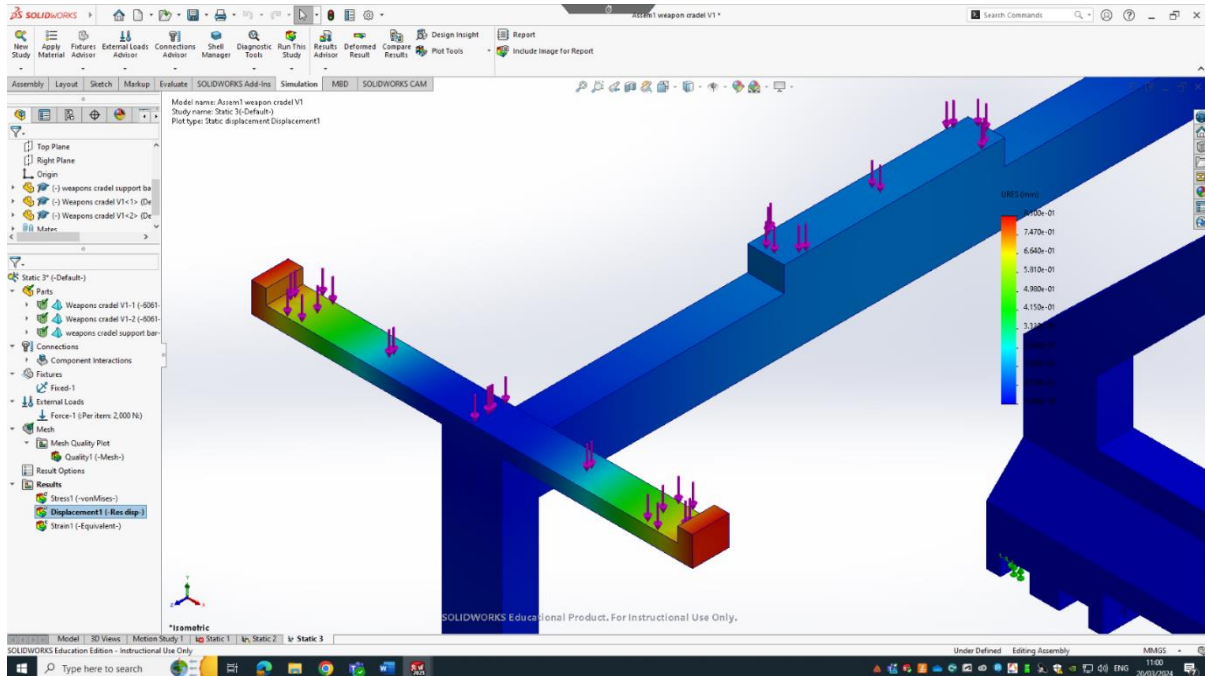
Showing Deformation

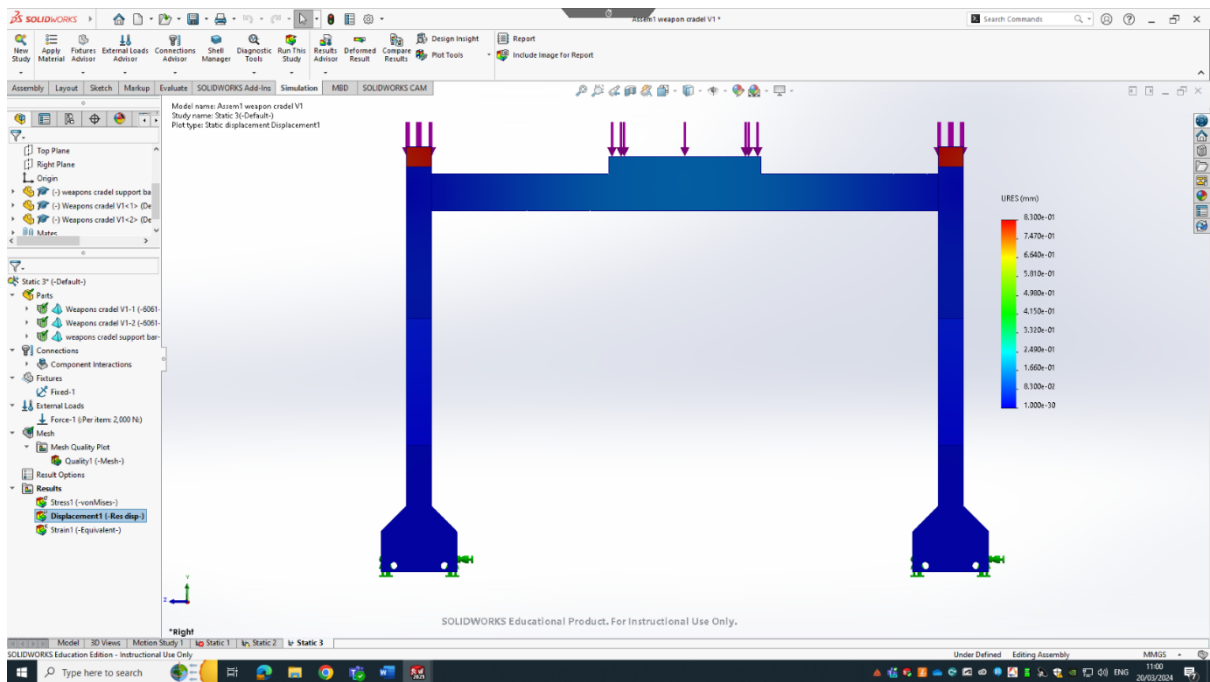
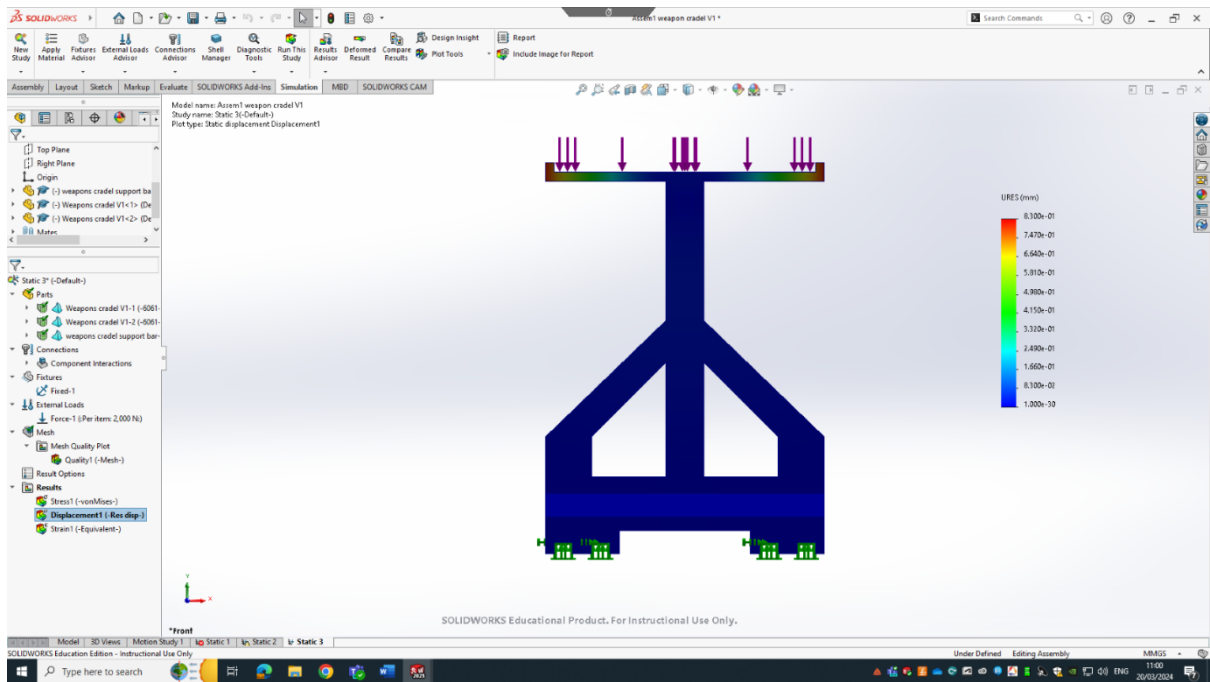


Appendix C

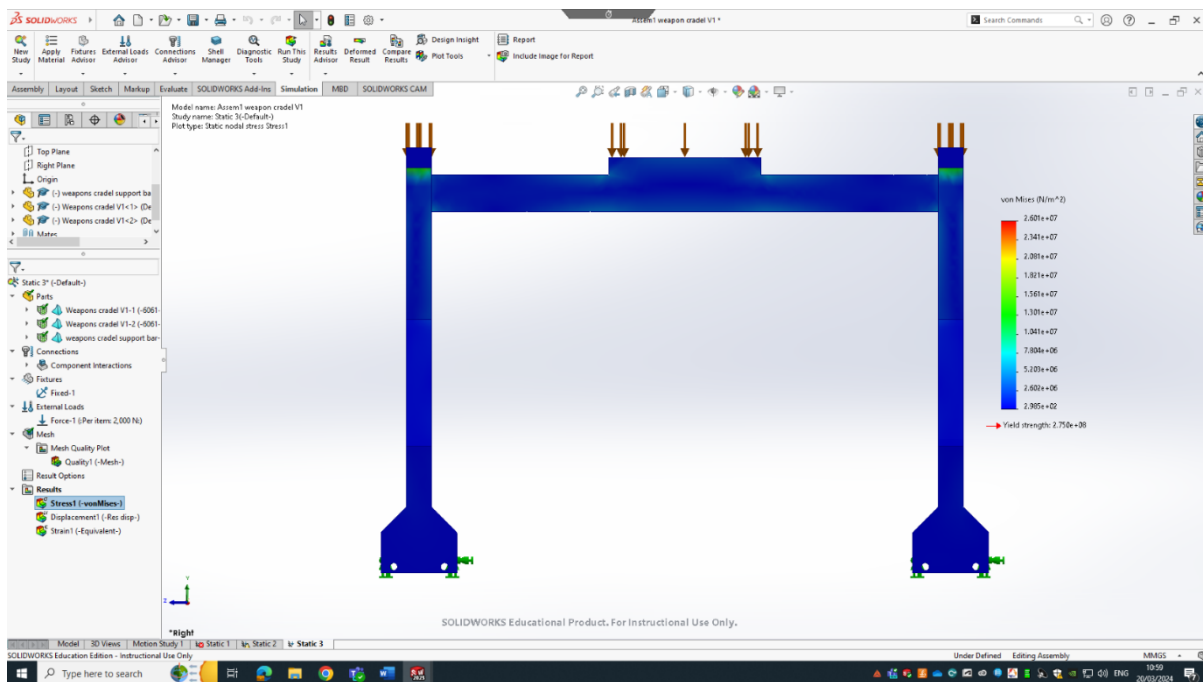
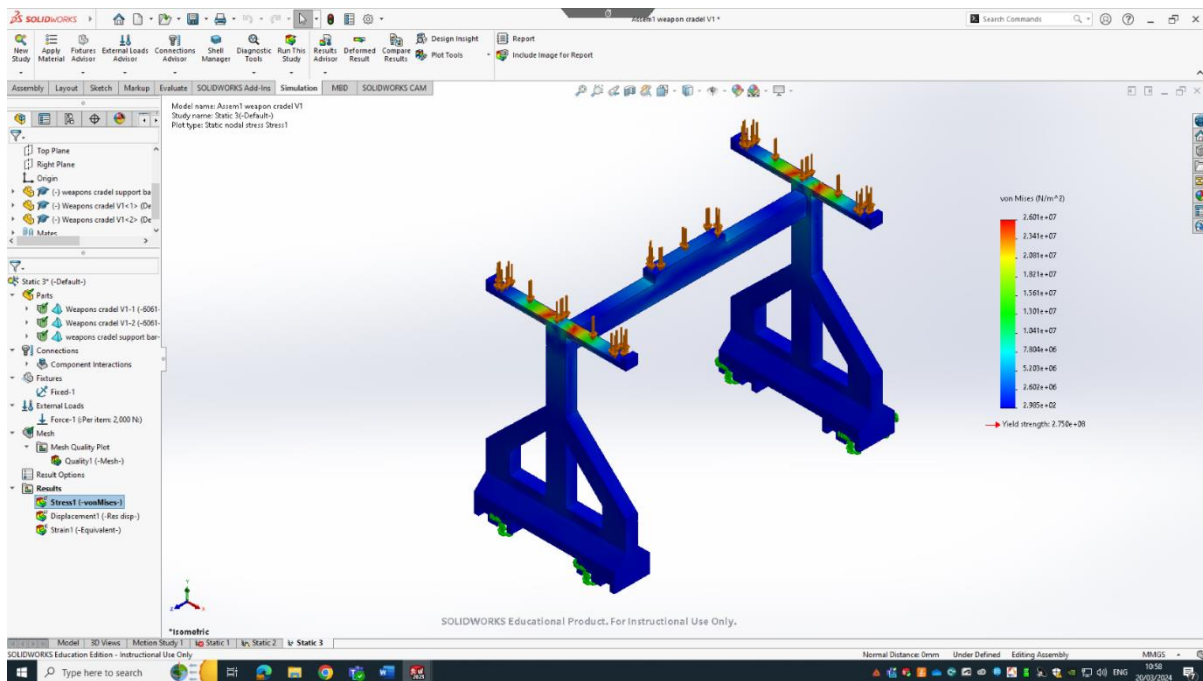
Assembly simulation in solidworks 2000N across all three contact points, mesh size Max 30mm min 2.67729mm

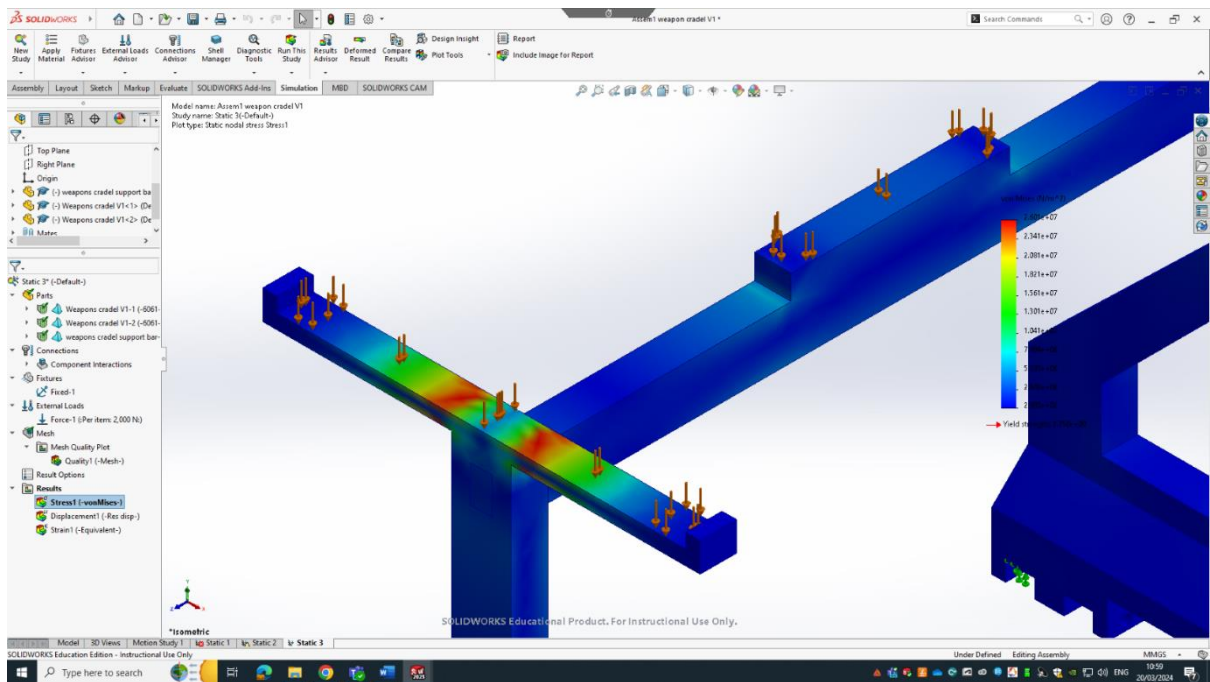
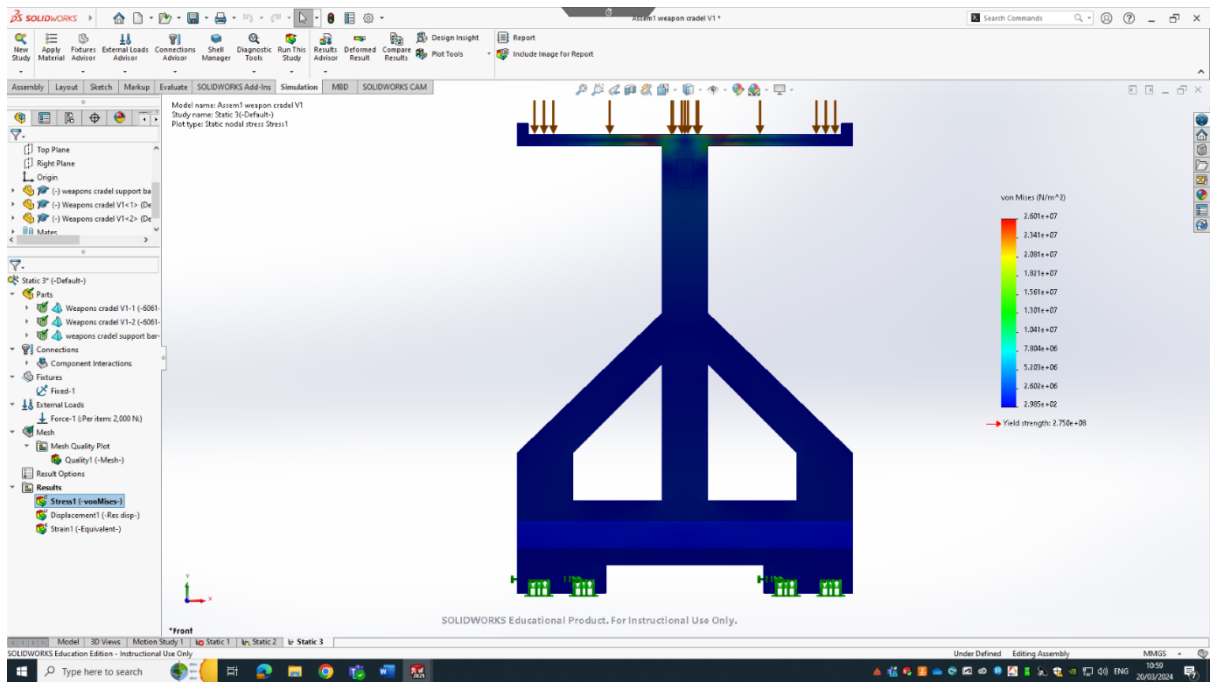
Showing Deformation



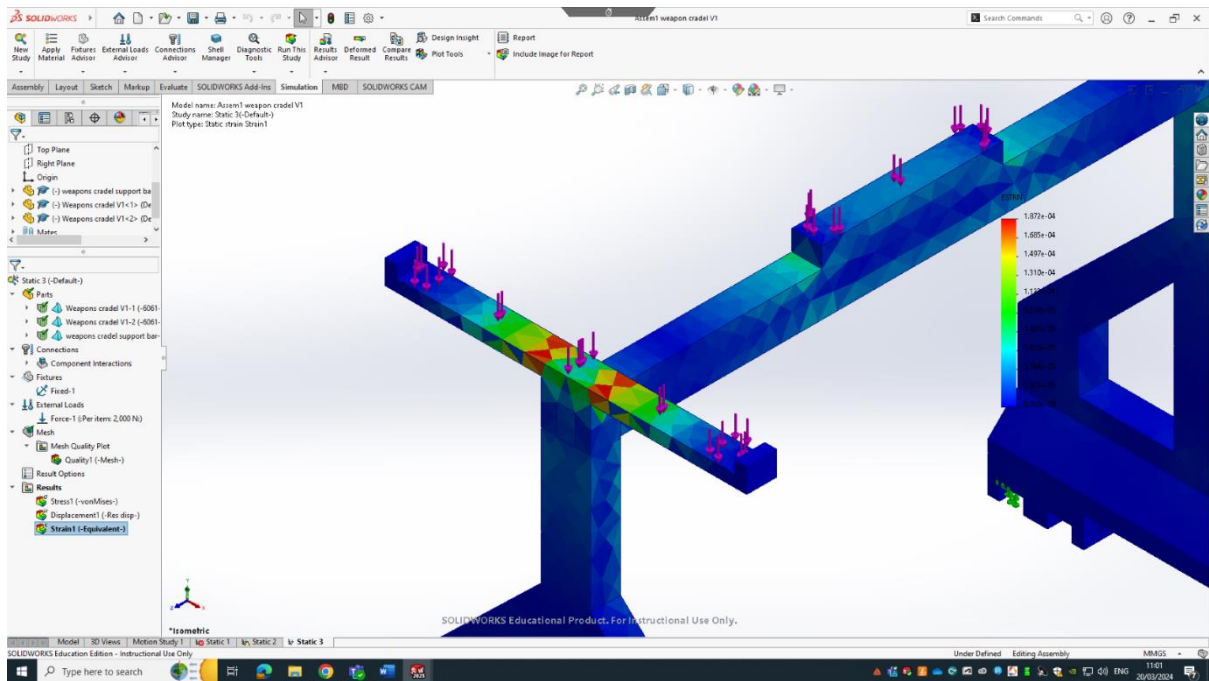
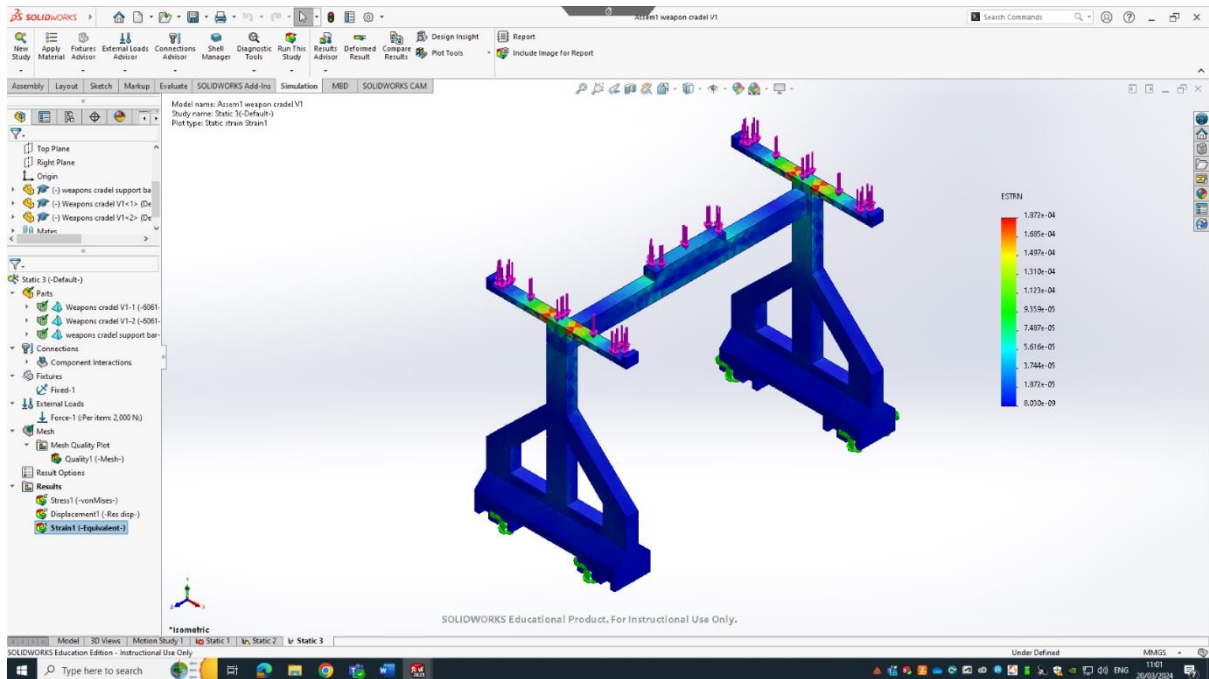


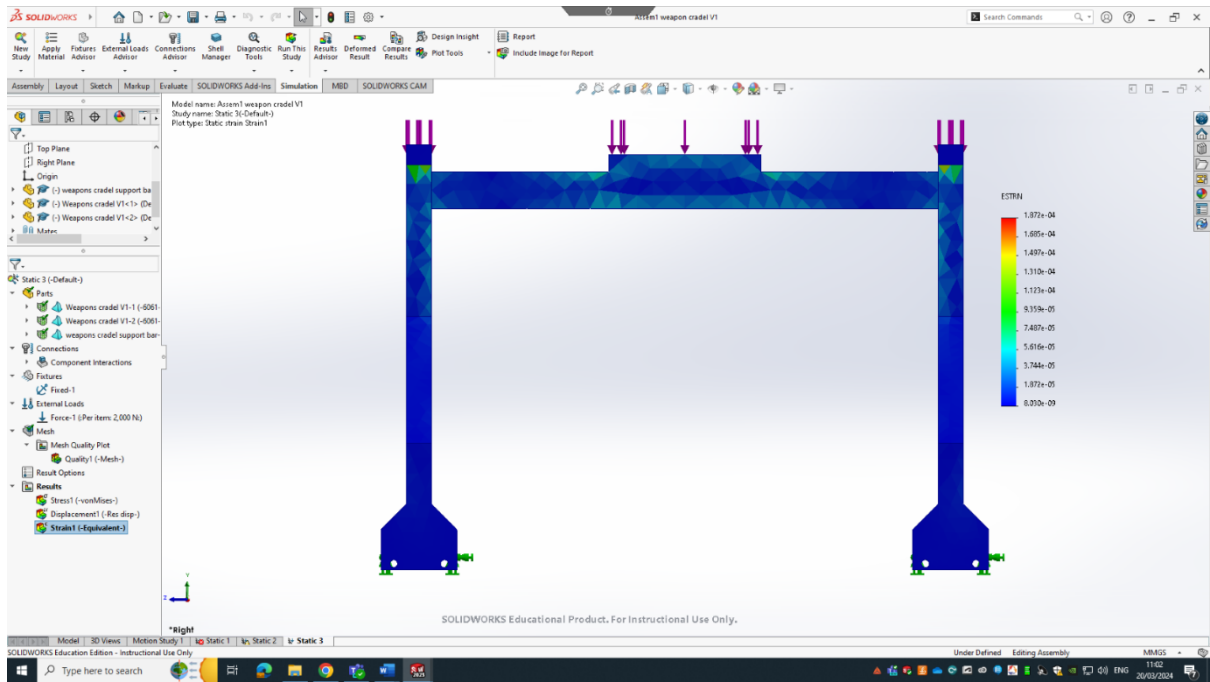
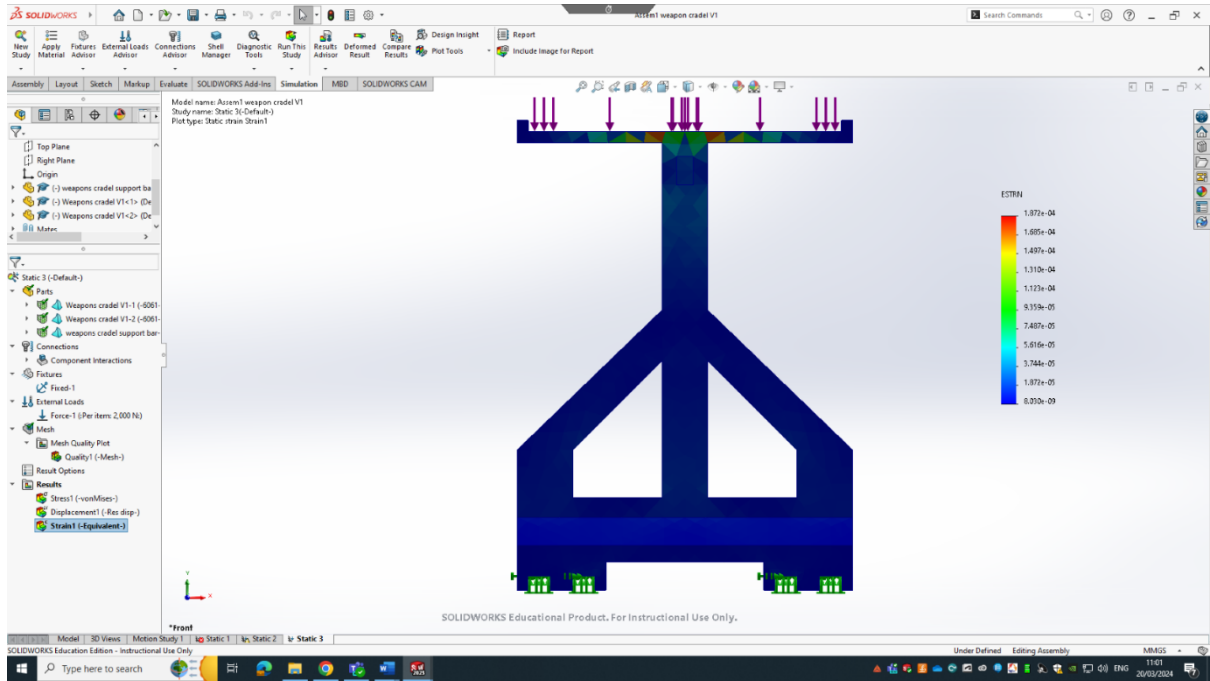
Showing Stress





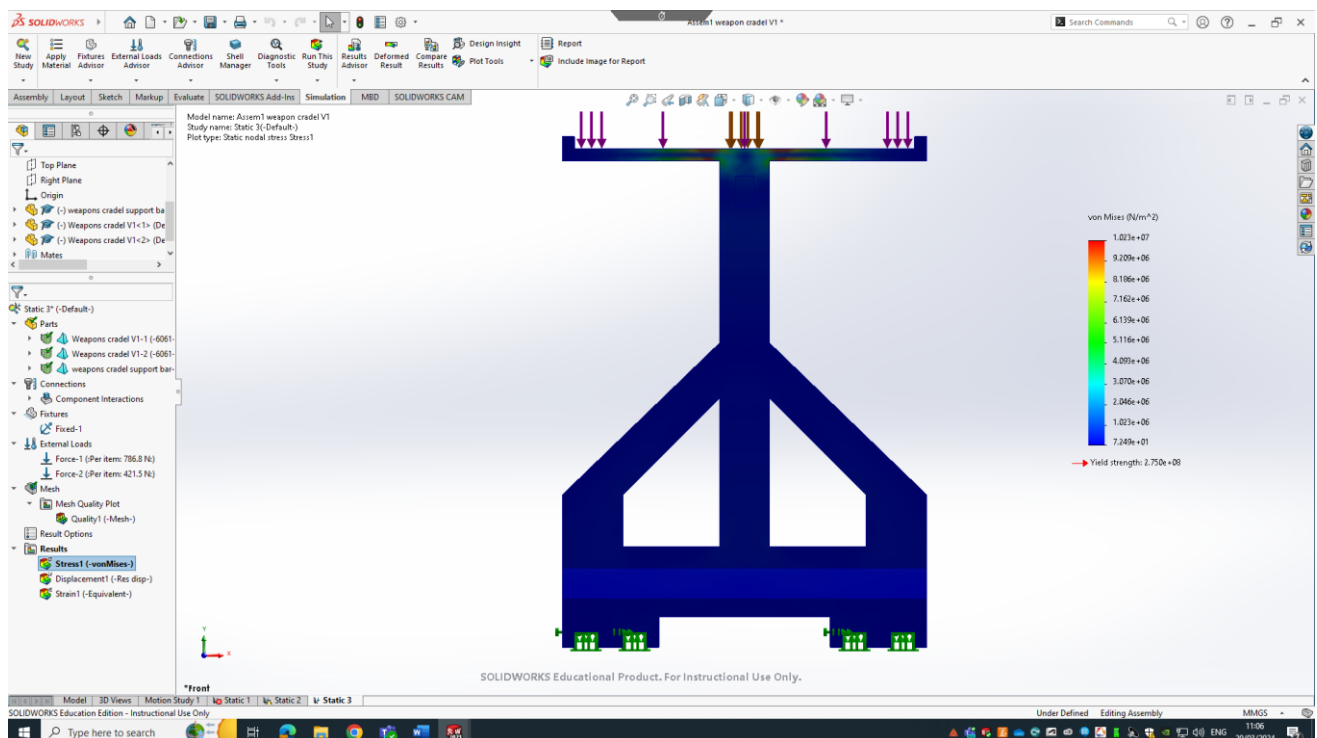
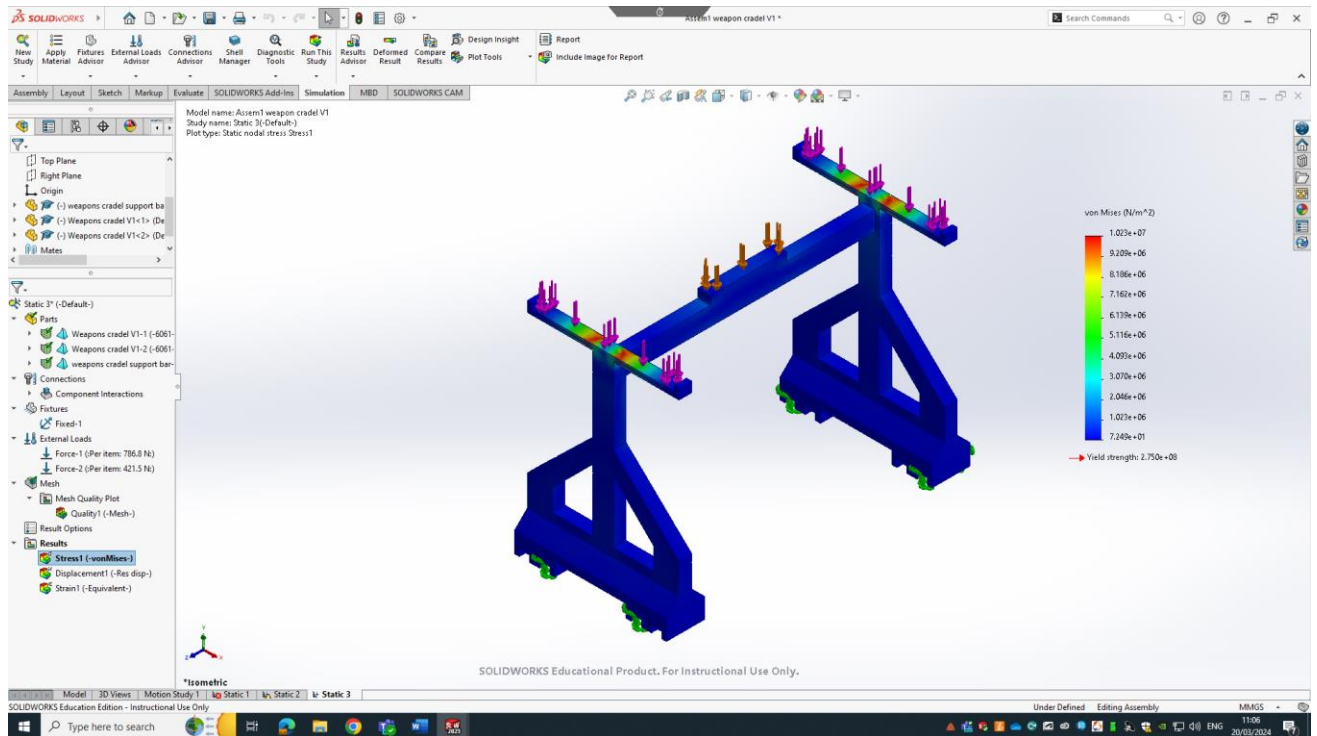
Showing Strain

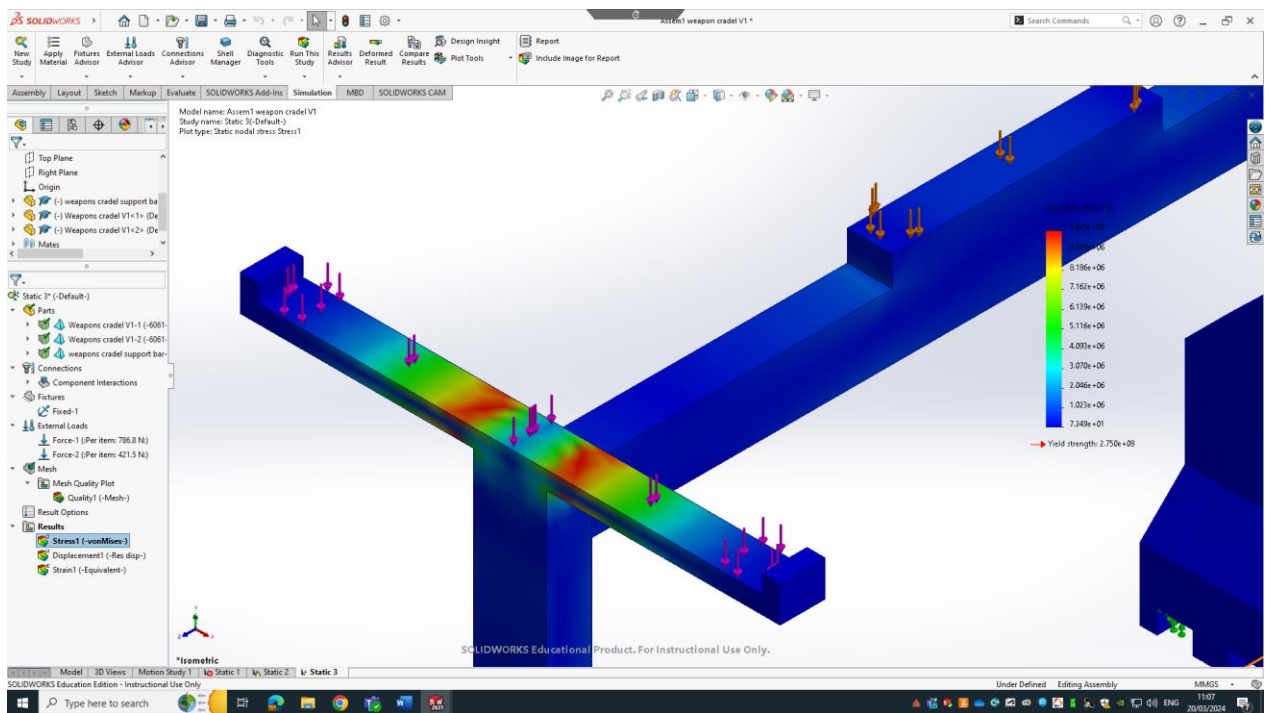
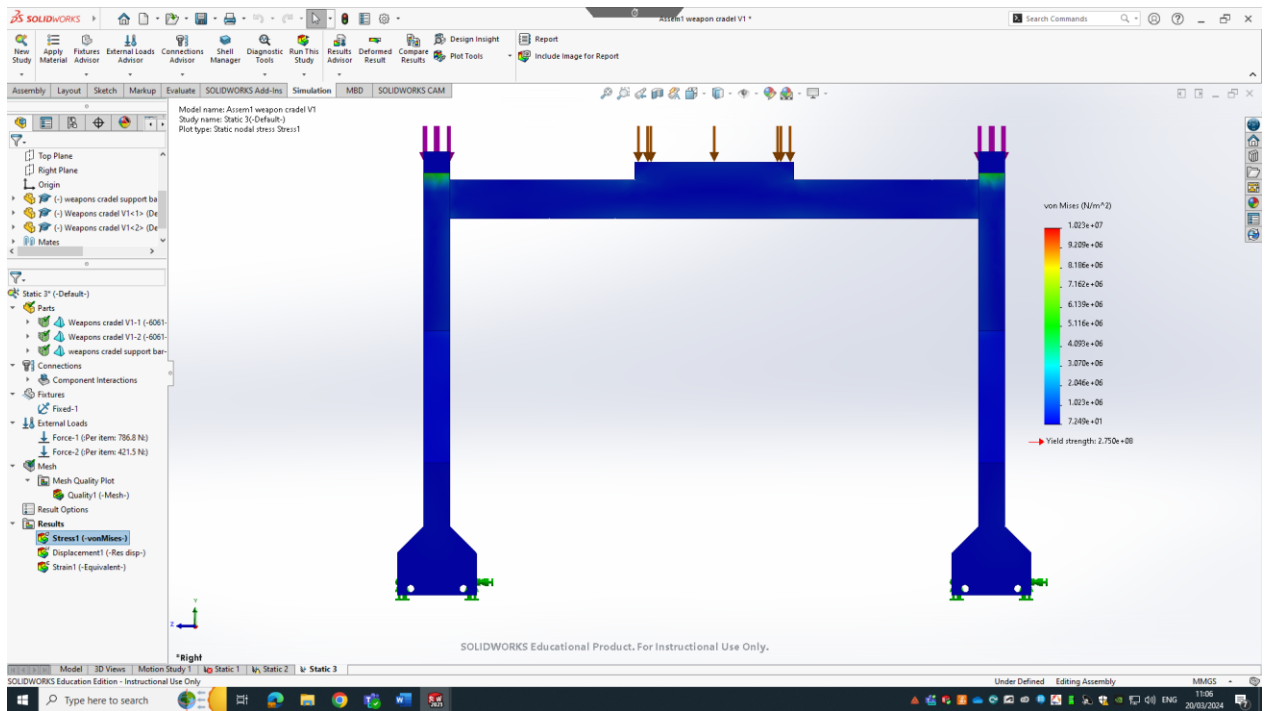




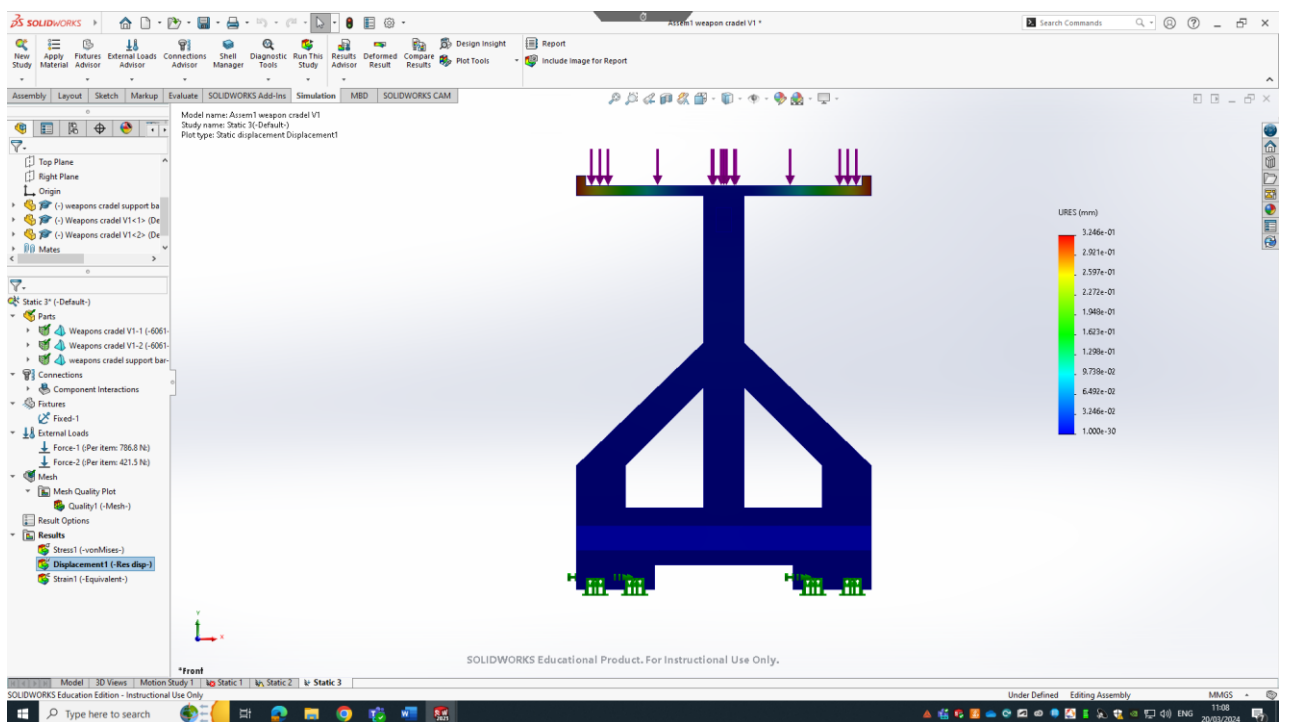
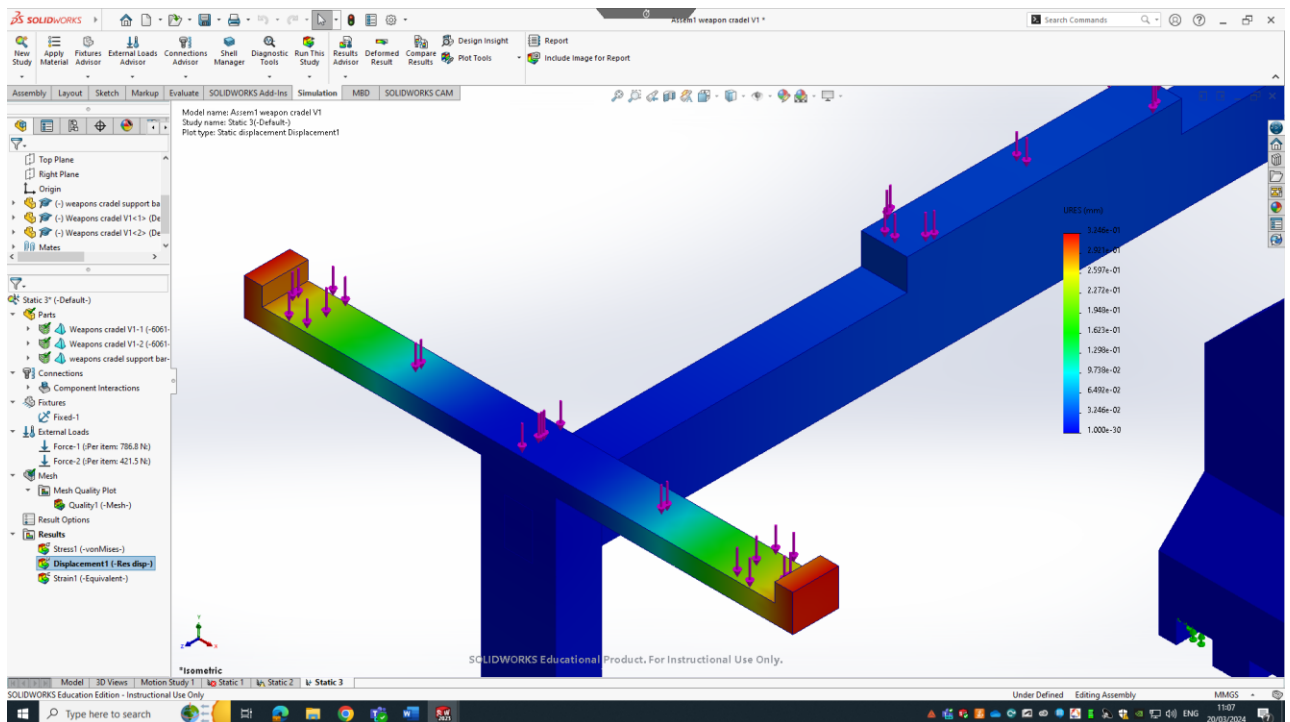
Assembly simulation in solidworks representative load of 786.8n & 421.5n across the three contact points, mesh size Max 30mm min 2.67729mm

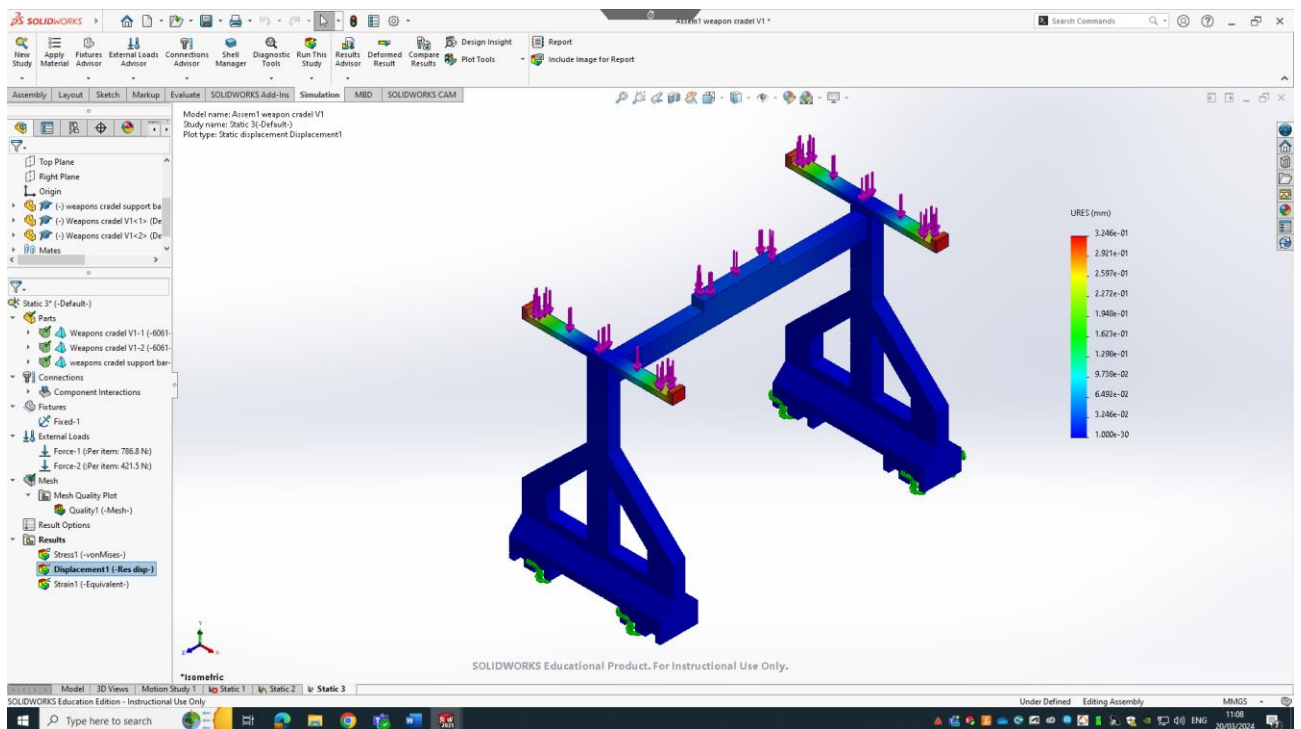
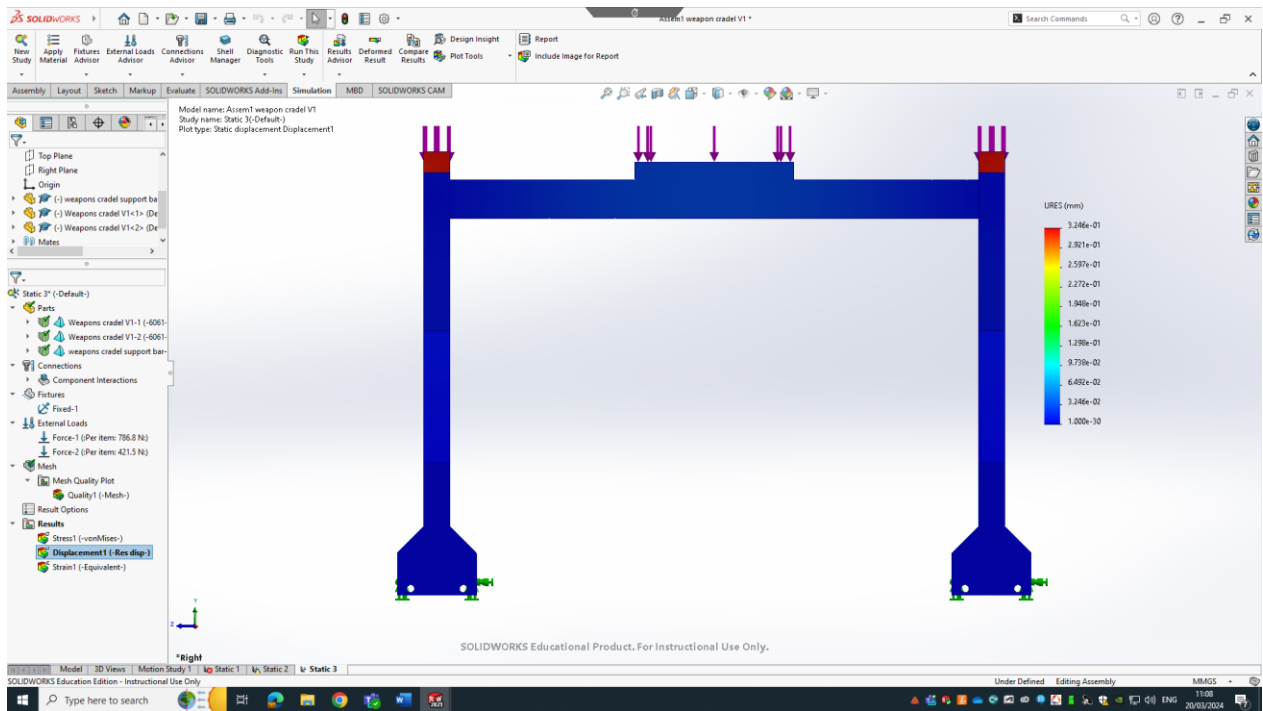
Showing Stress



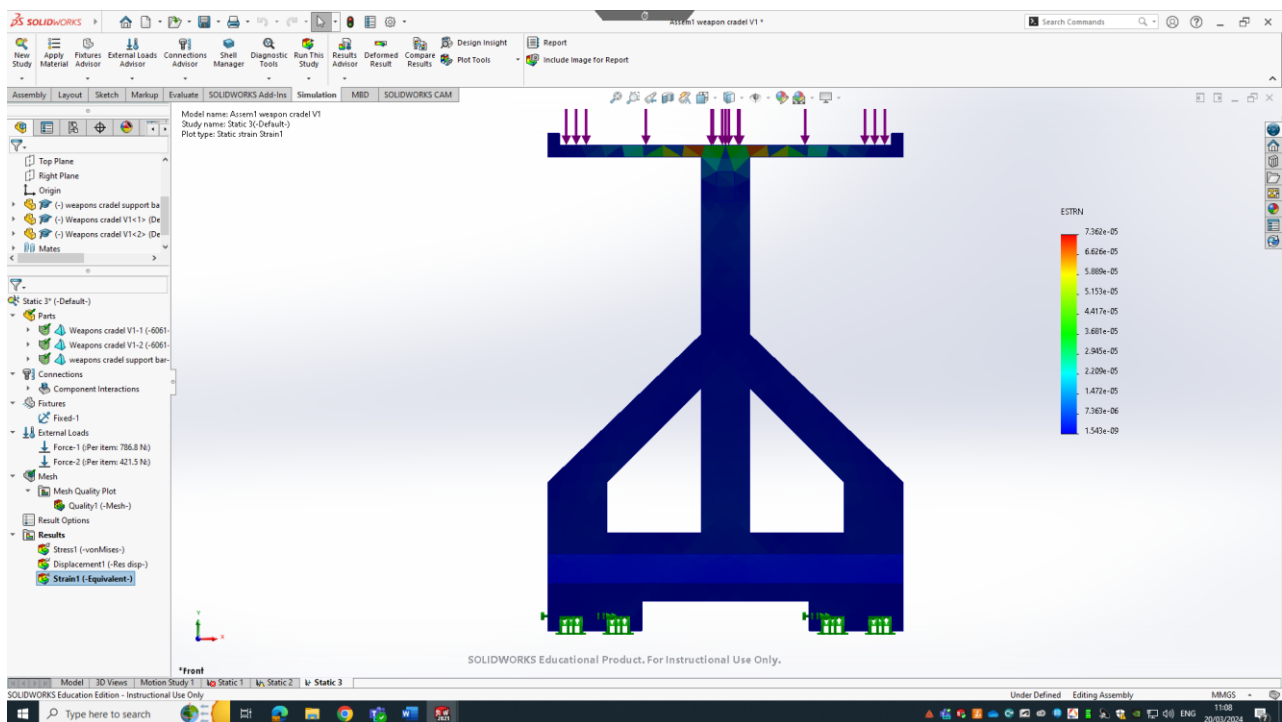
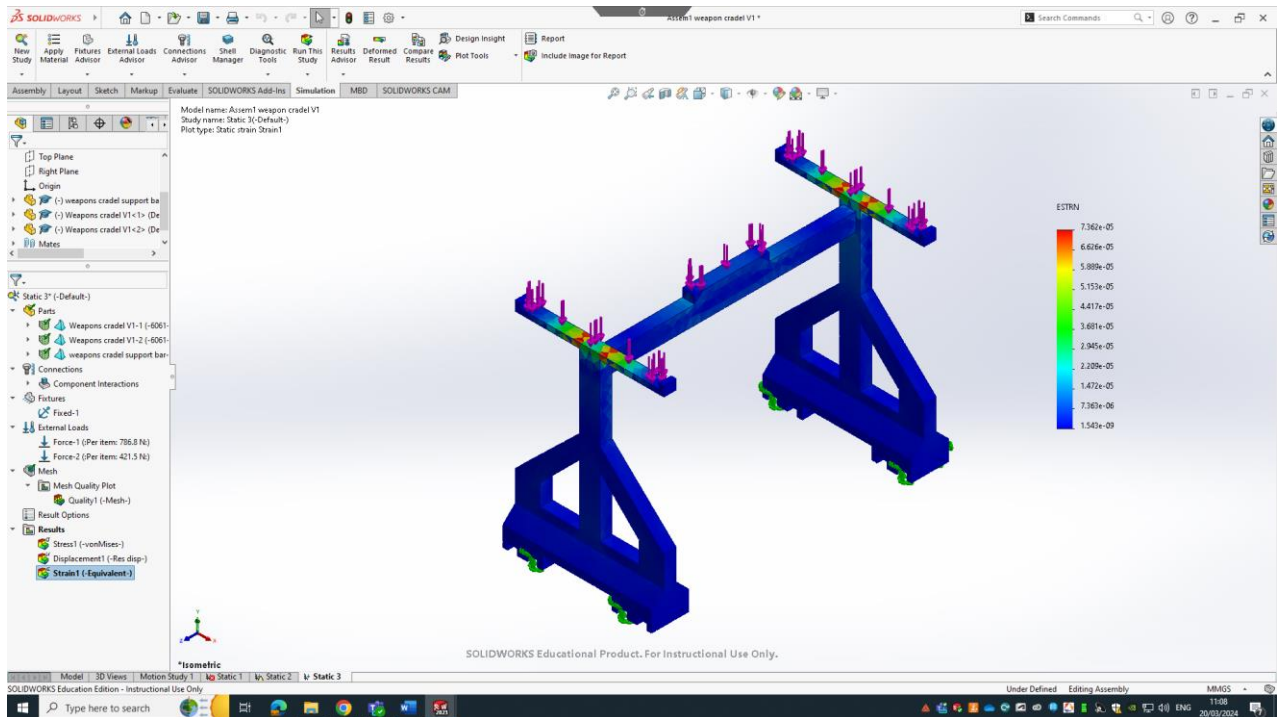


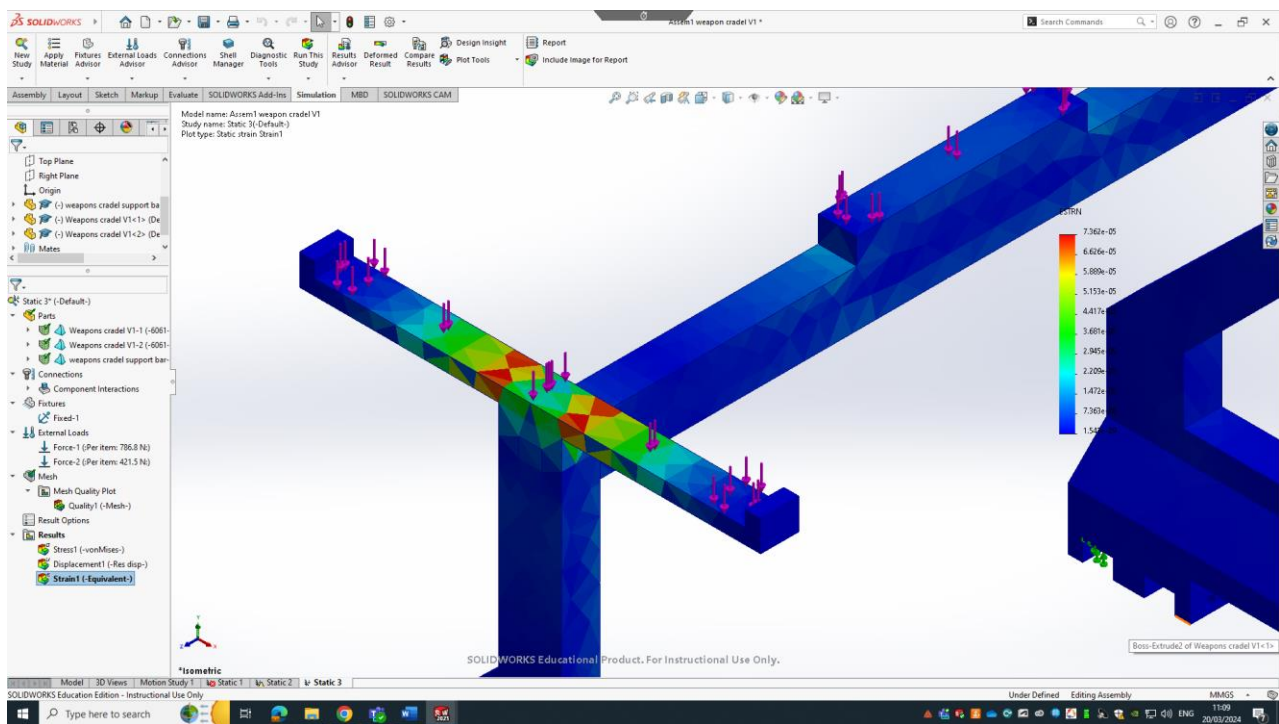
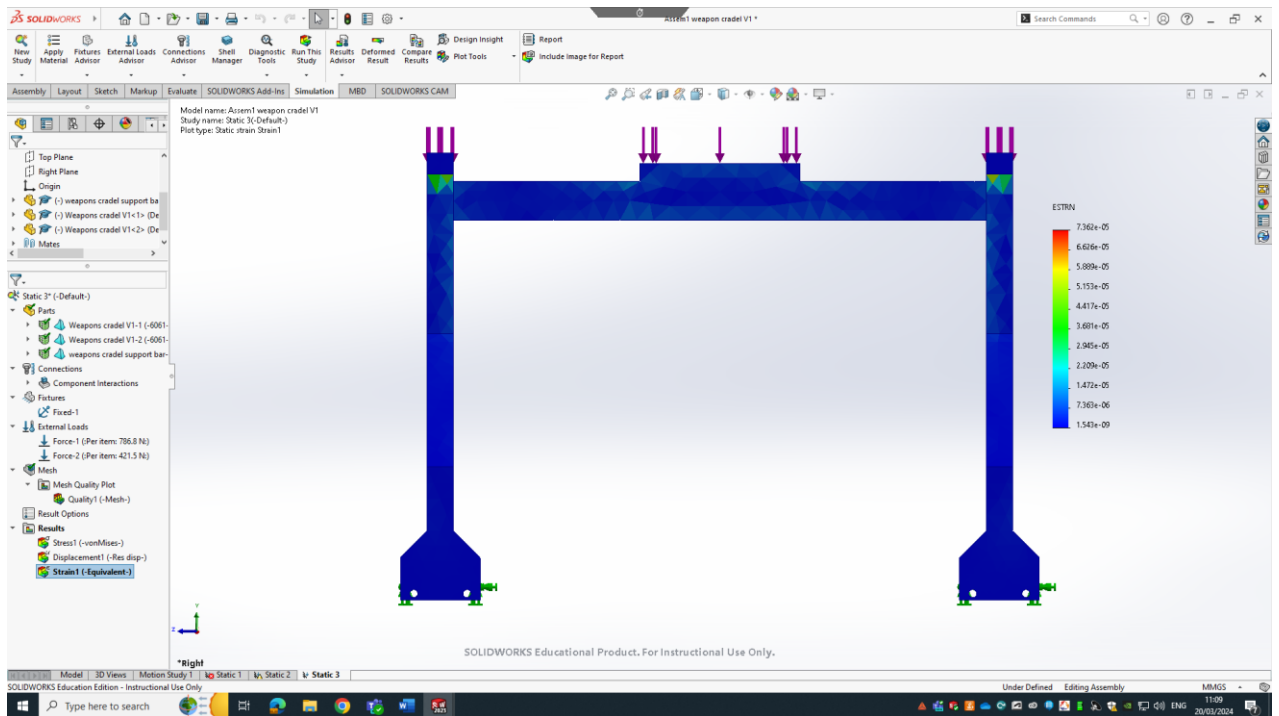
Showing Deformation





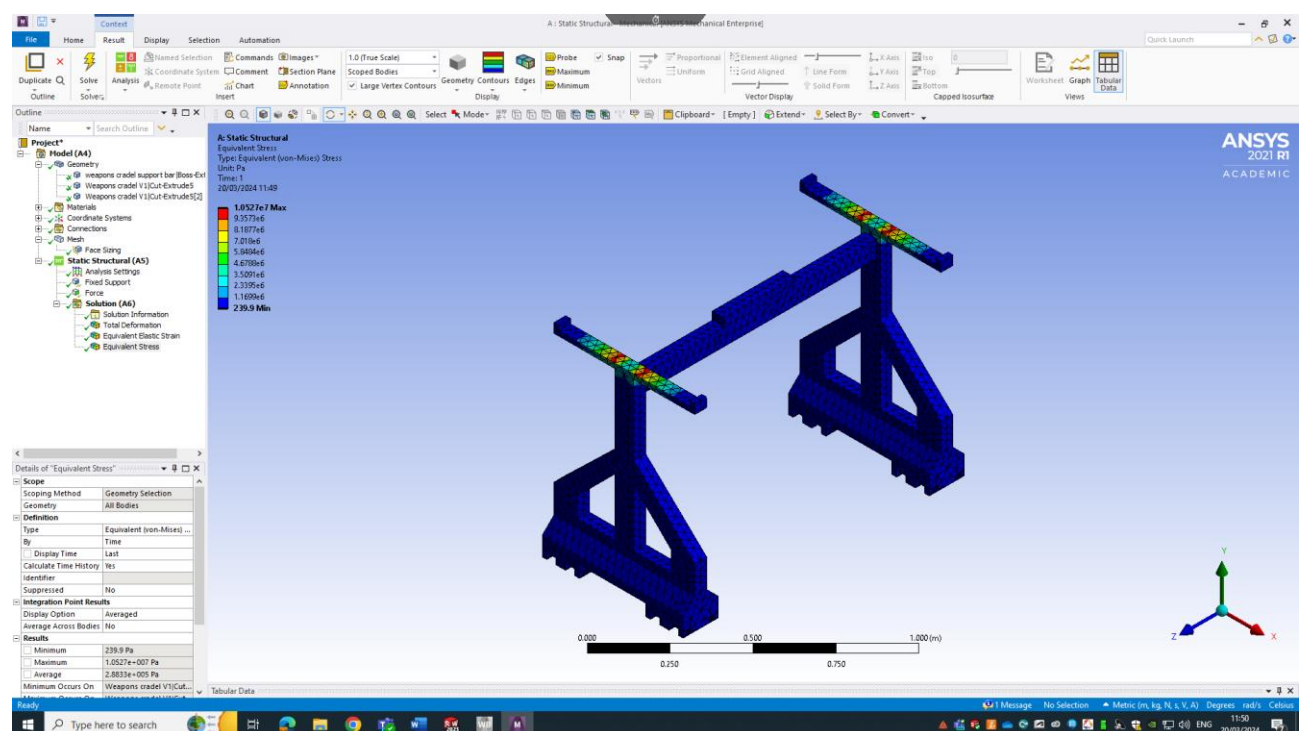
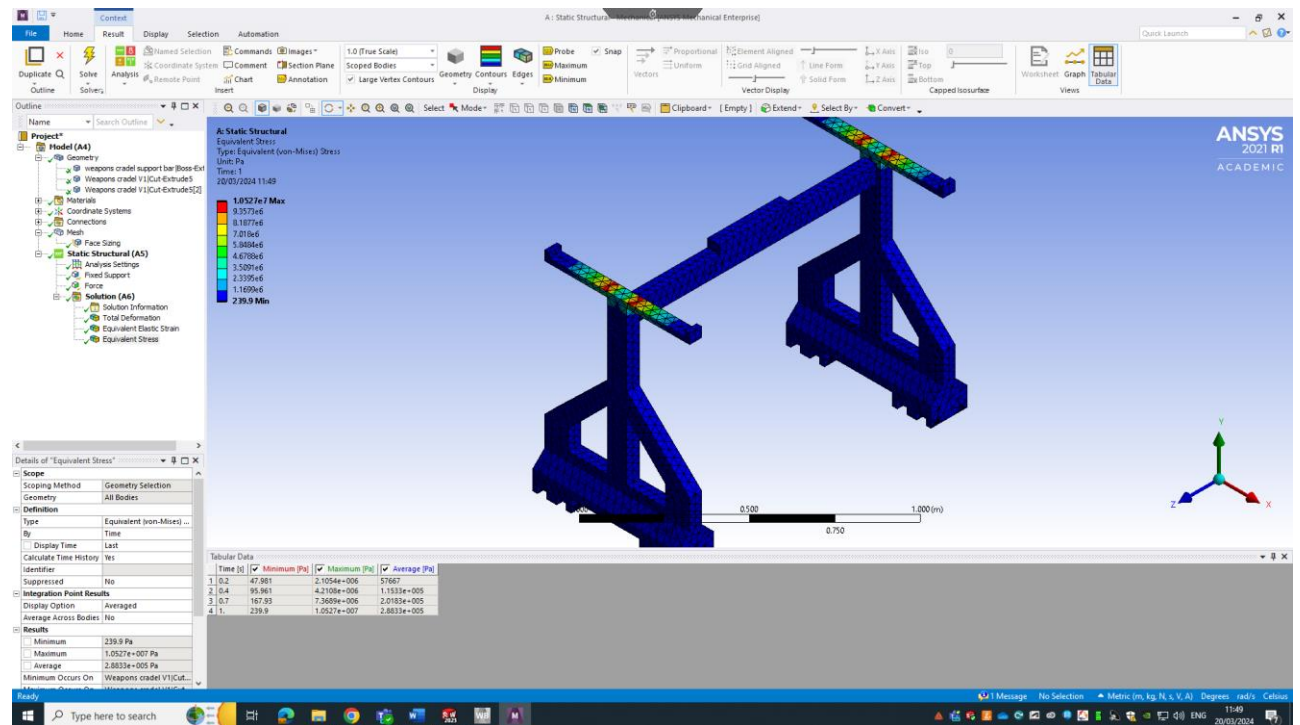
Showing Strain

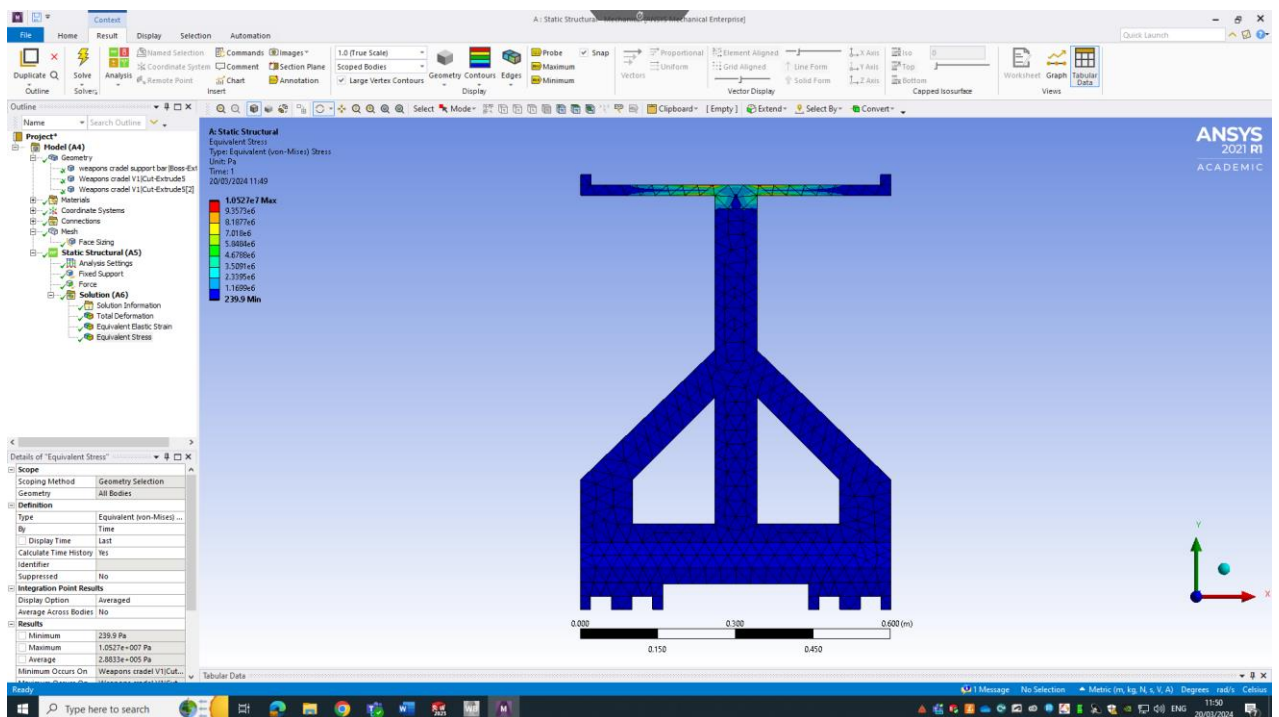
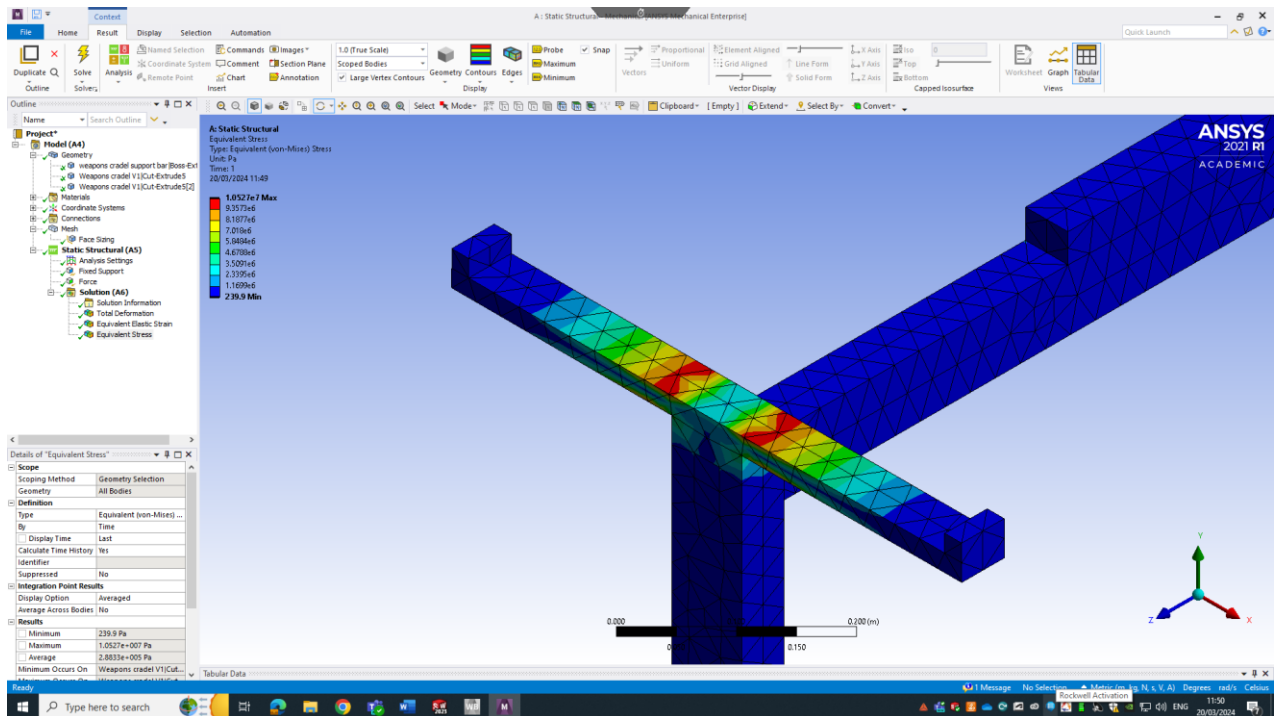


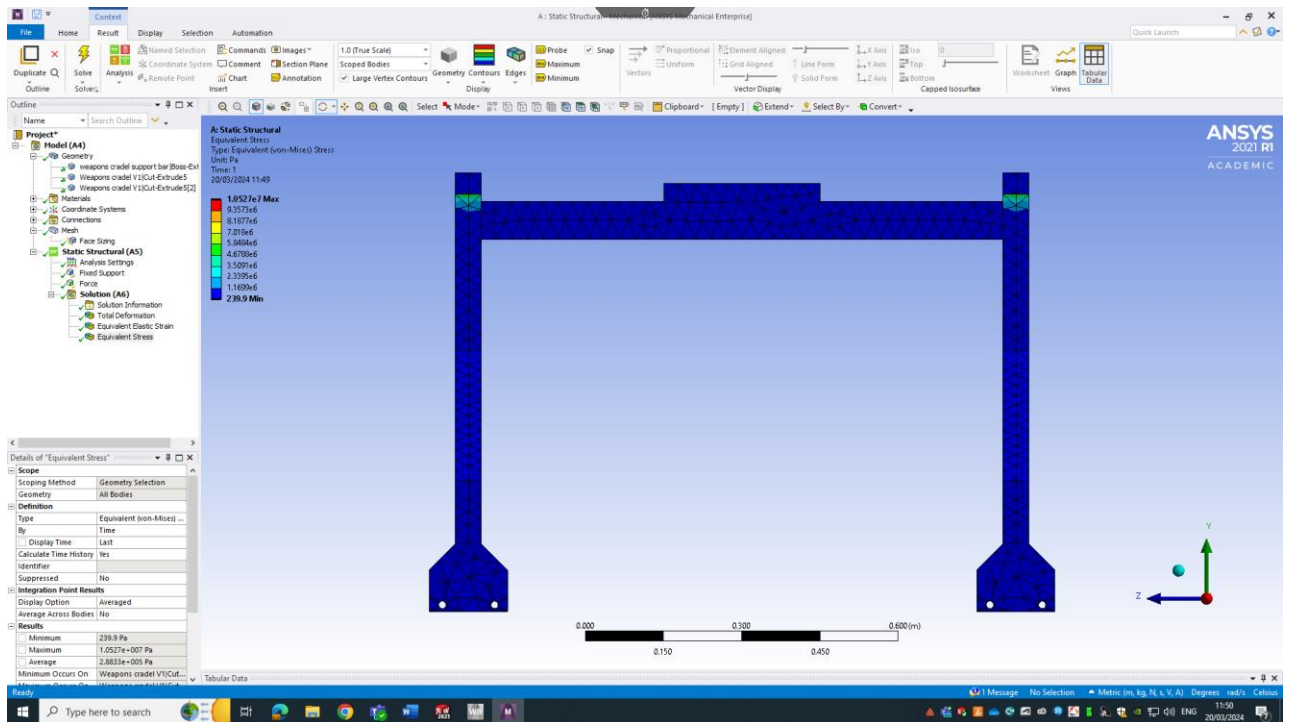


Assembly V1 simulation in Ansys representative load of 2000N across the three contact points, mesh size 30mm

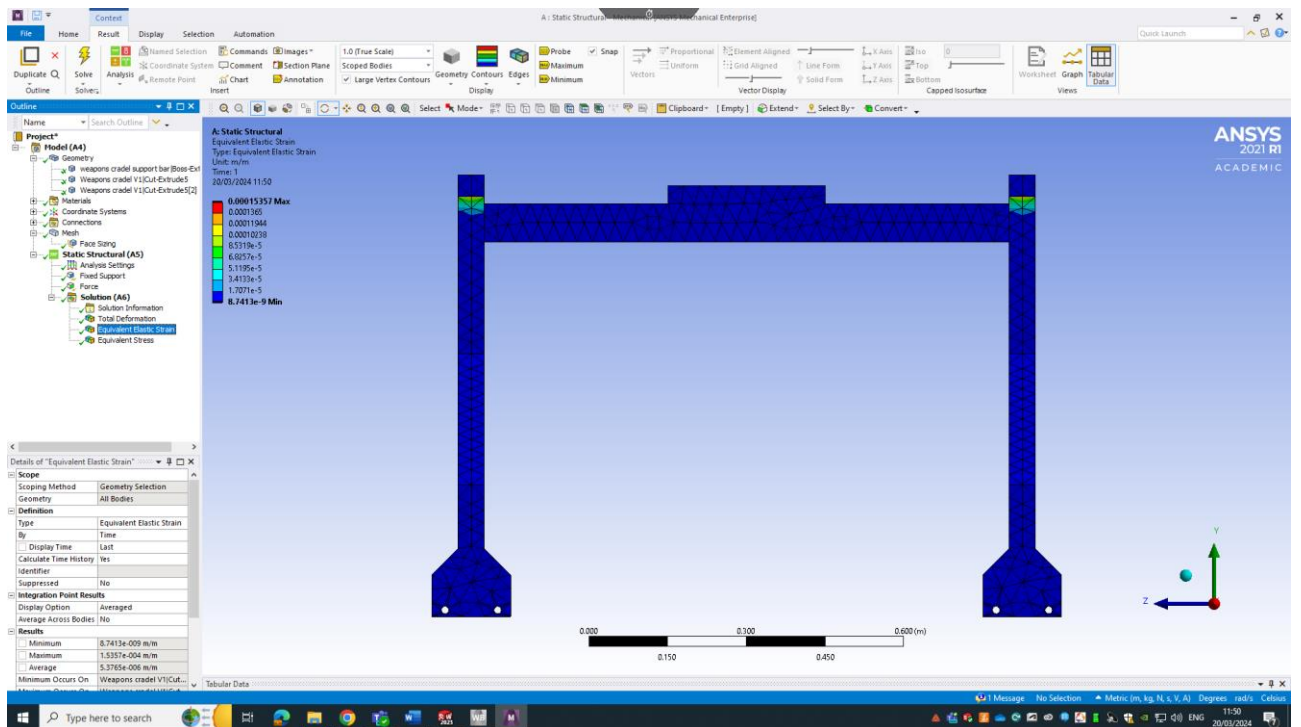
Showing Stress

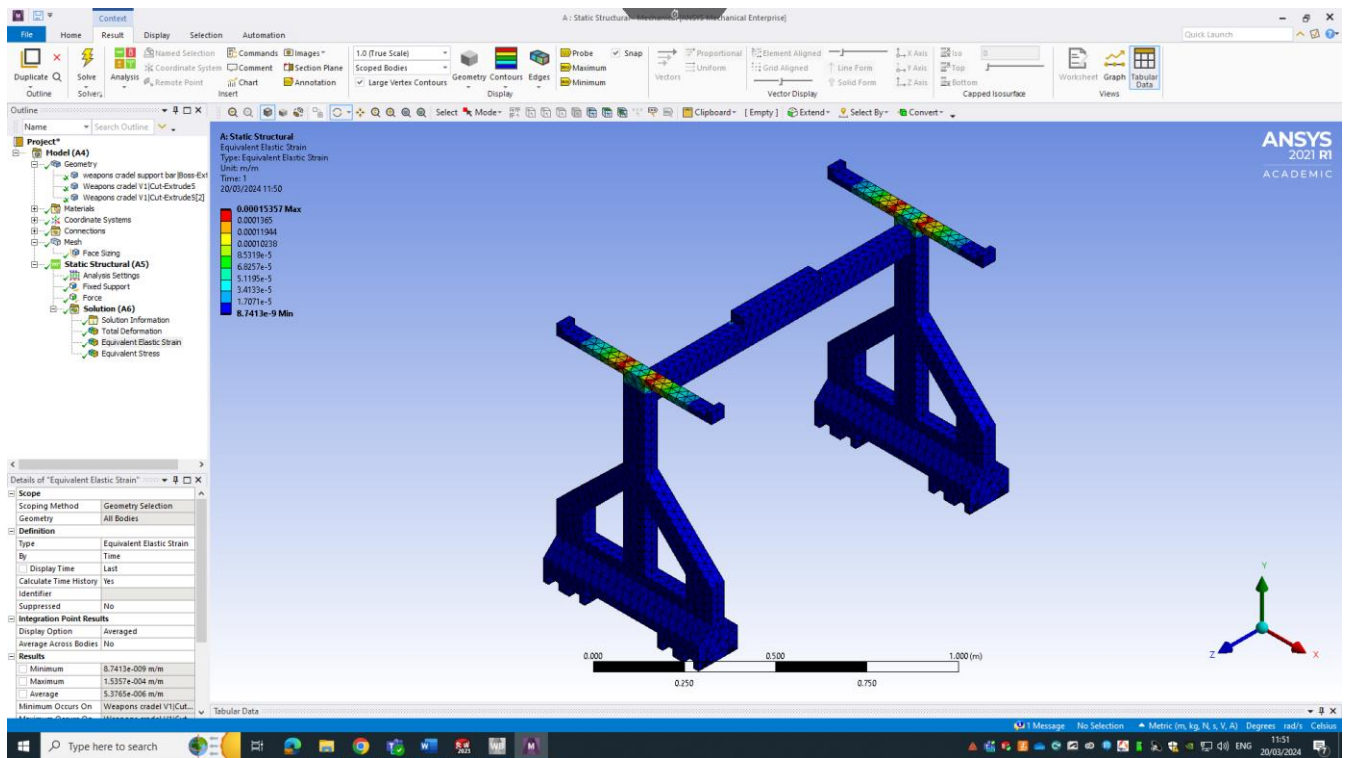
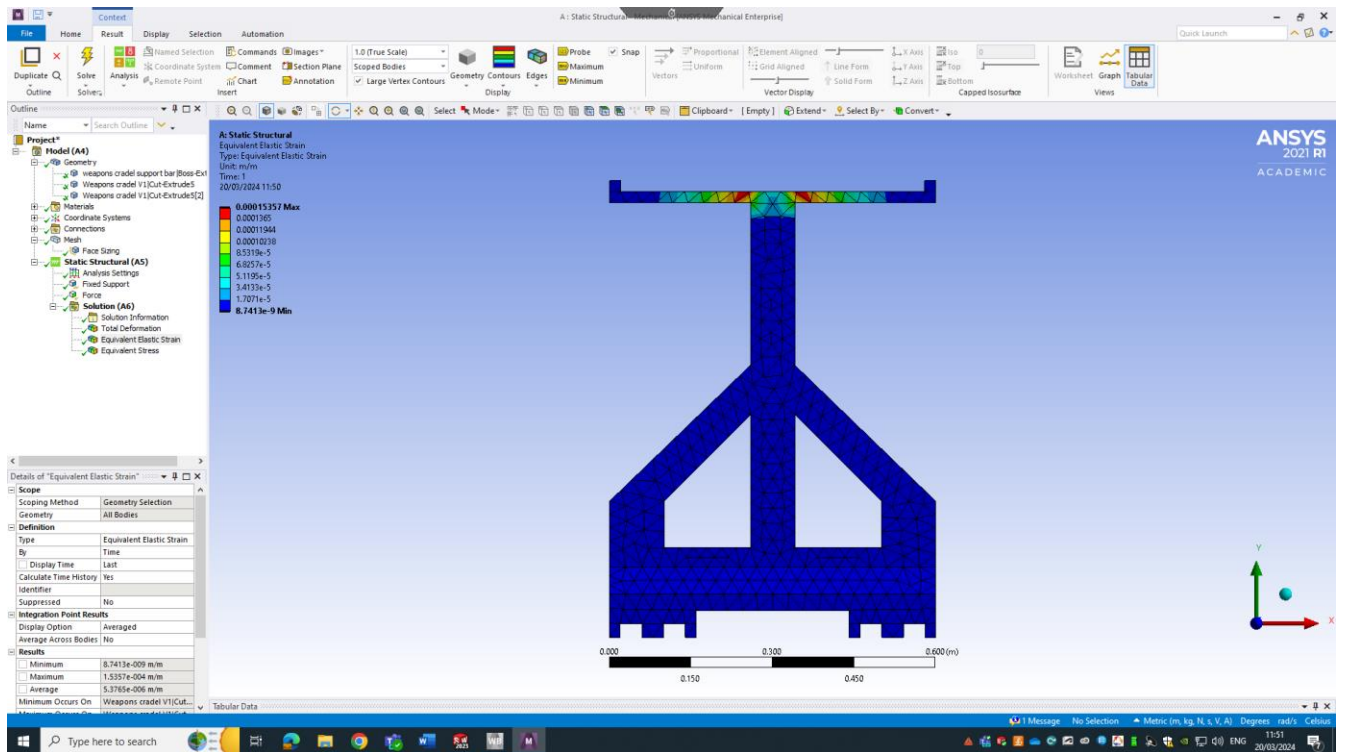


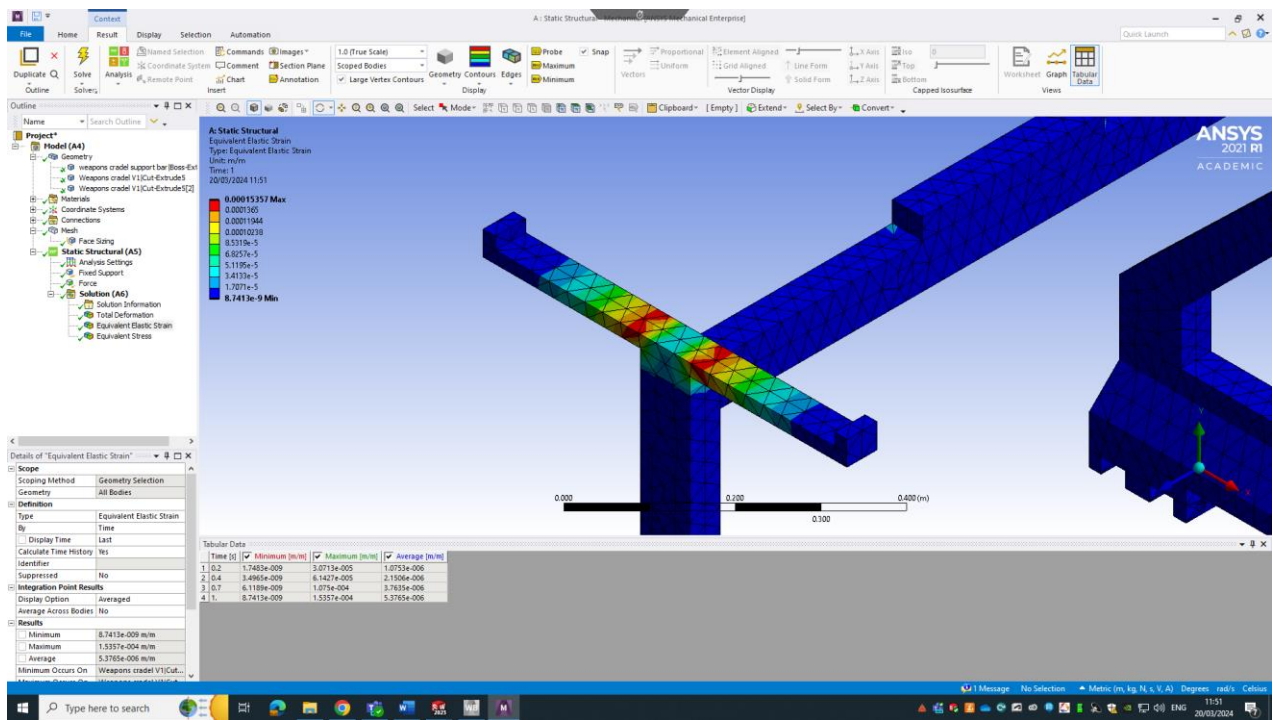
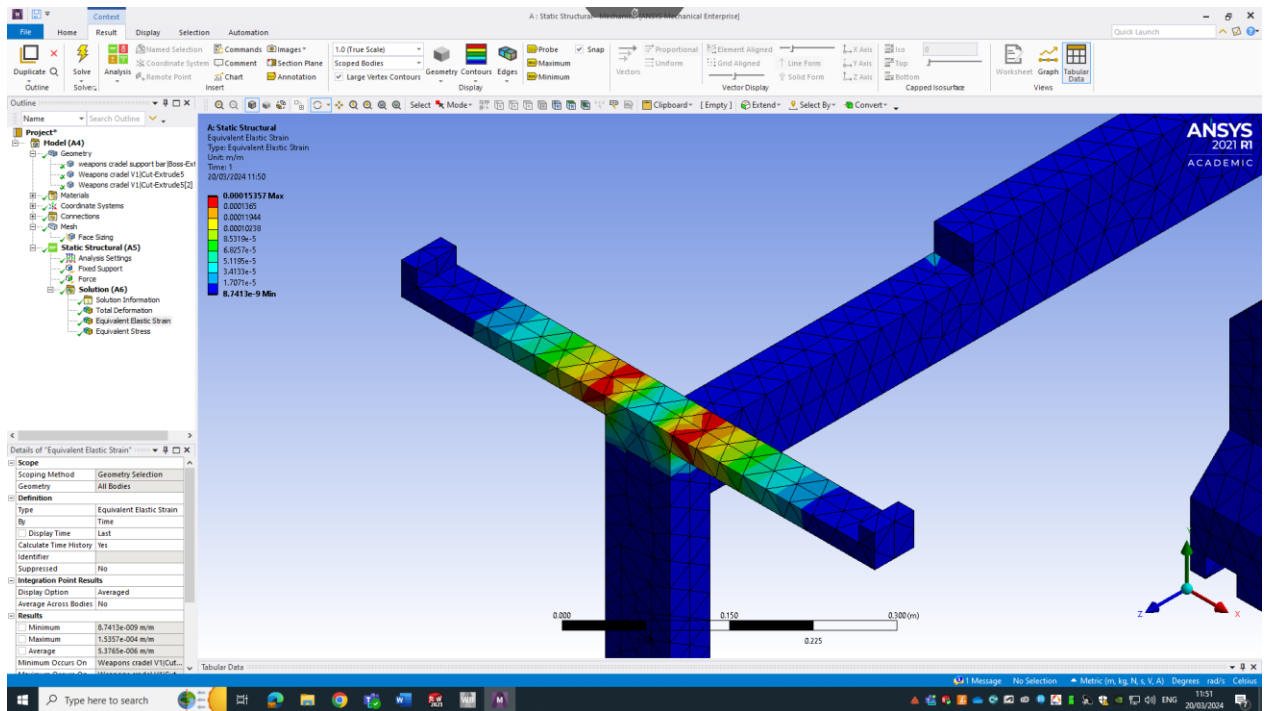




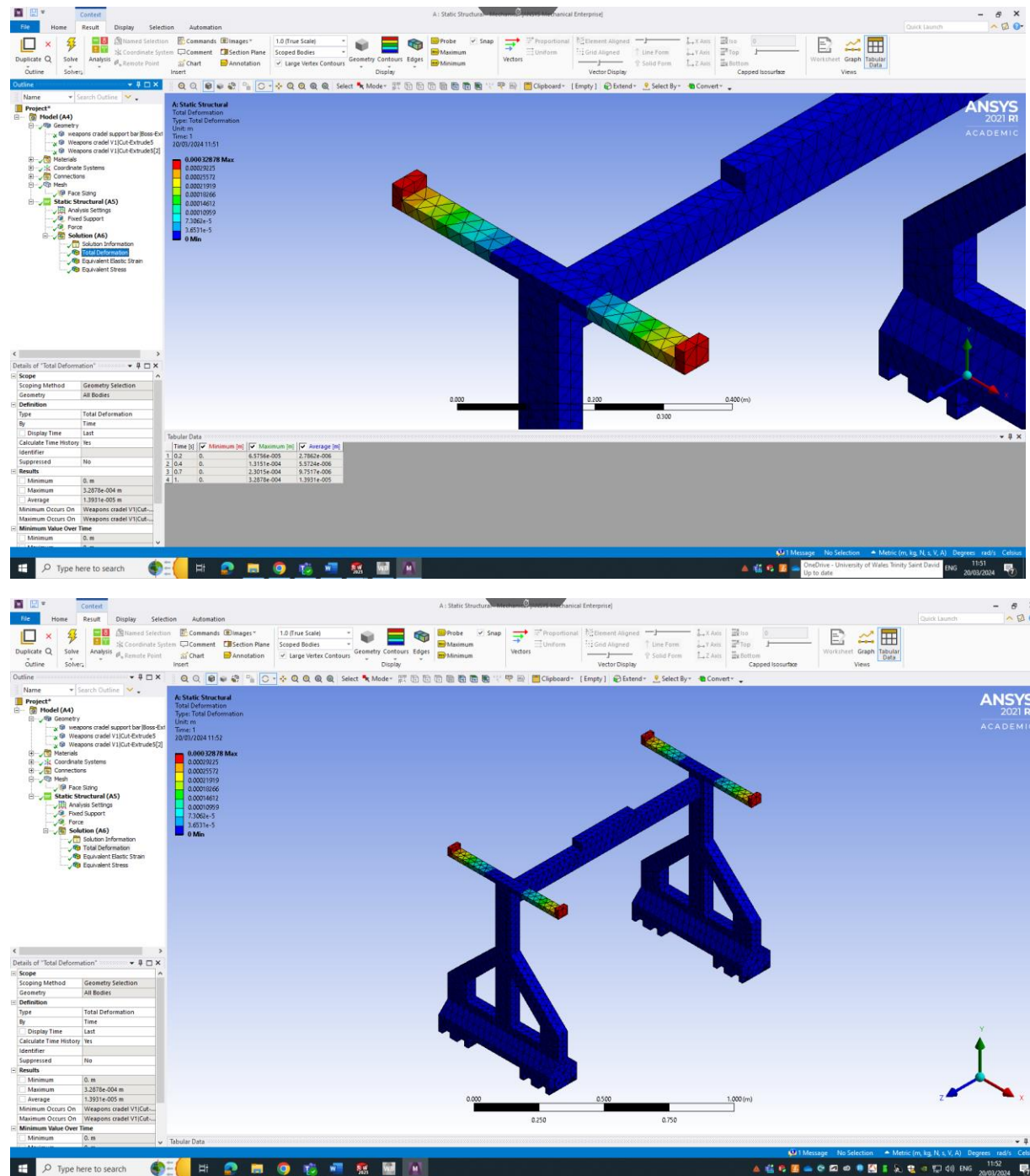
Showing Strain





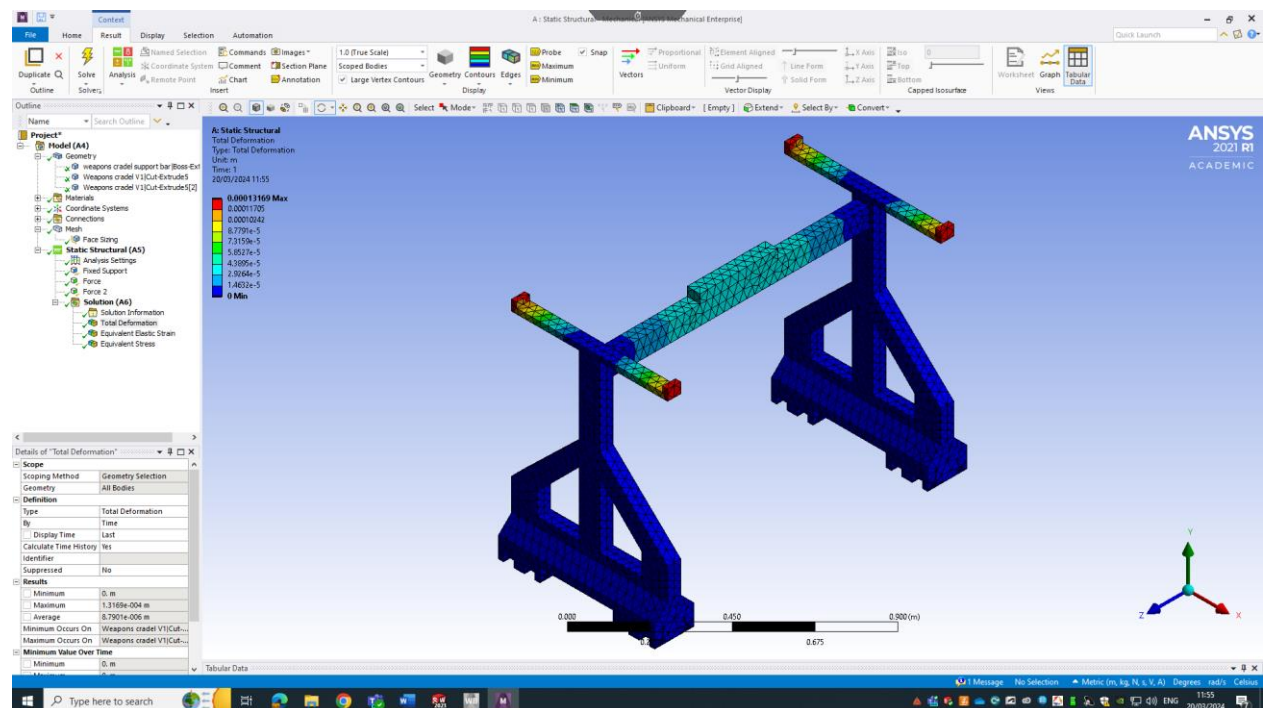
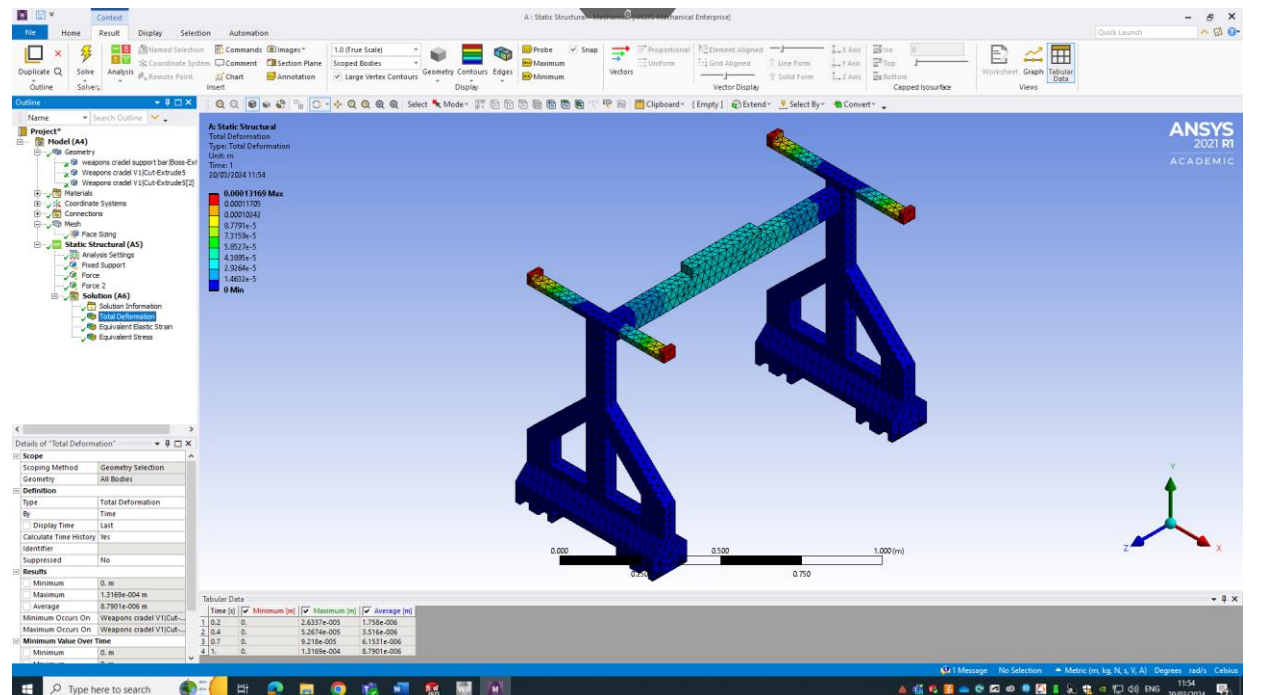


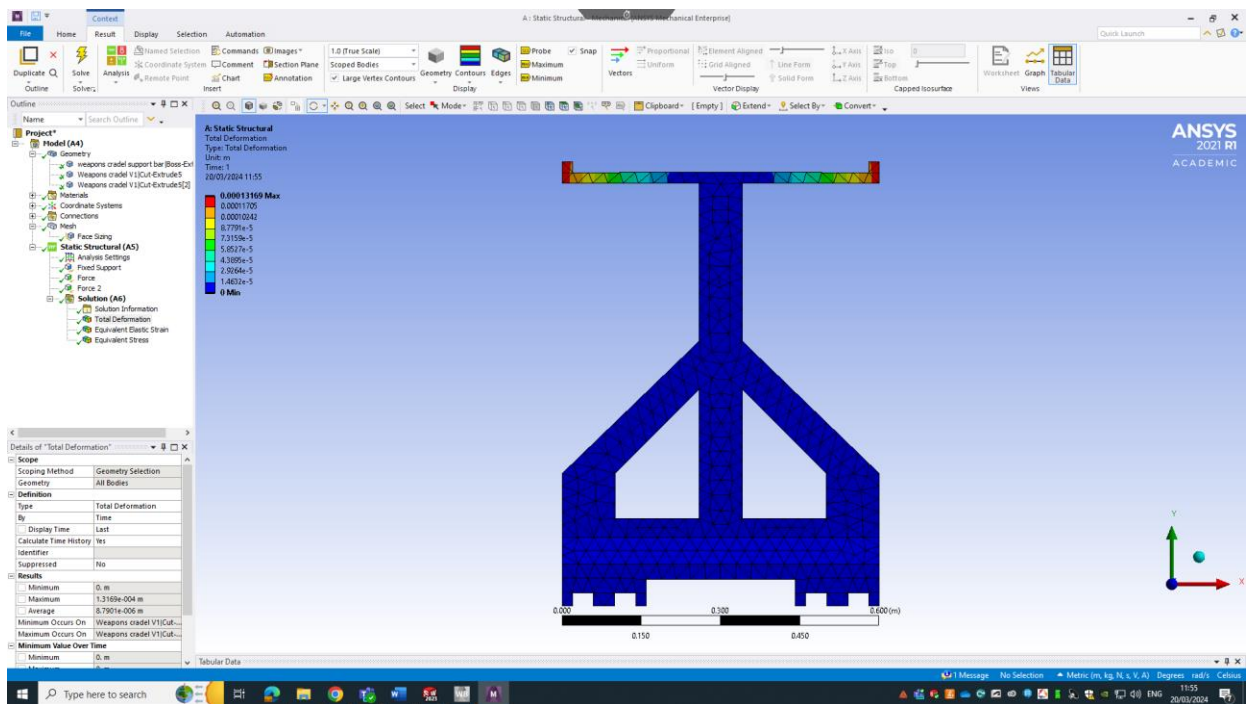
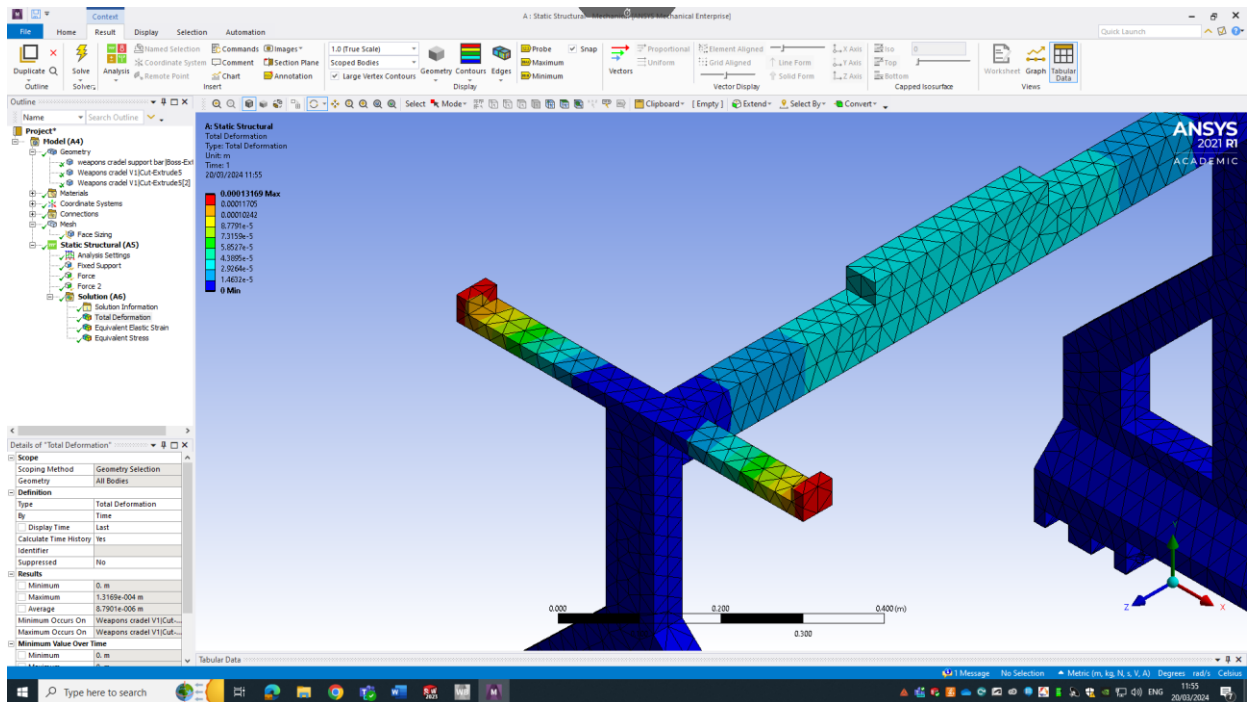
Showing Deformation

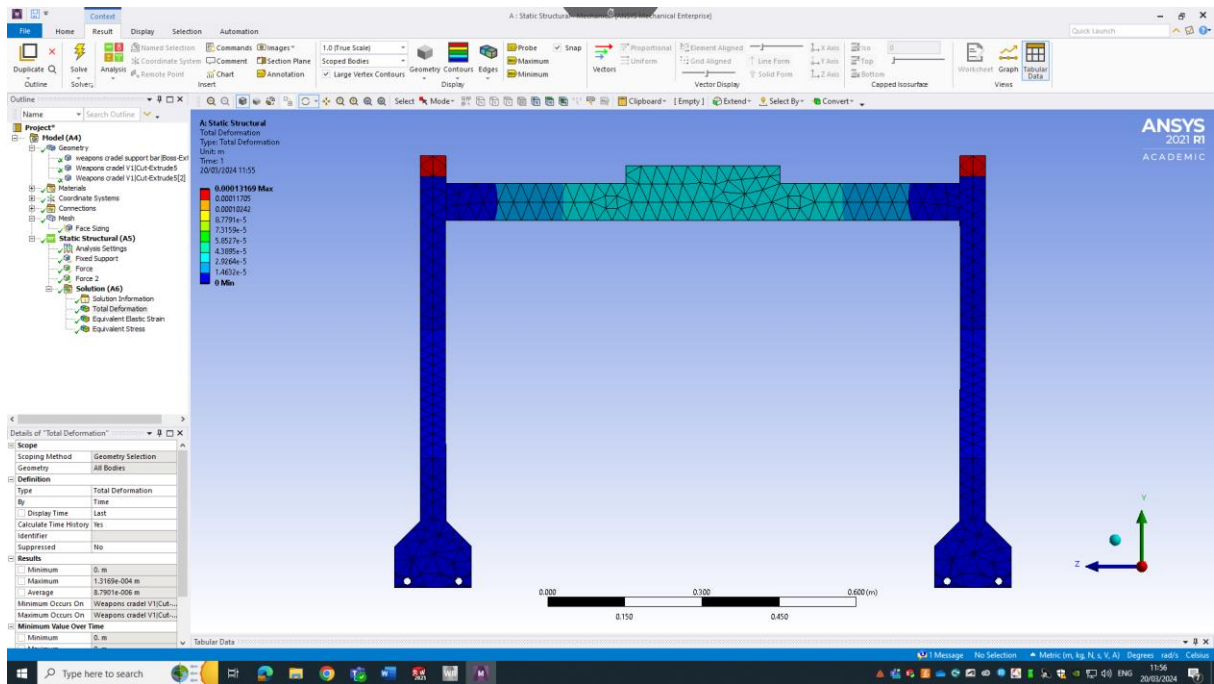


Assembly V1 simulation in Ansys representative load of 786.8n & 421.5n across the three contact points, mesh size 30mm

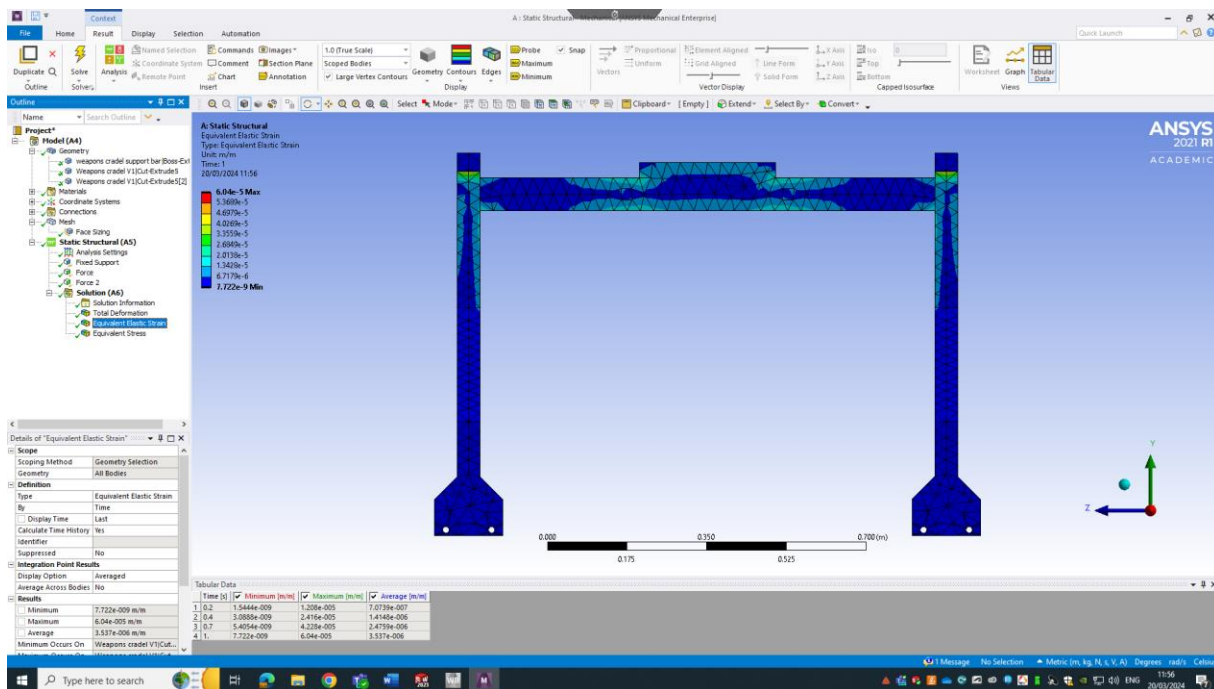
Showing Deformation

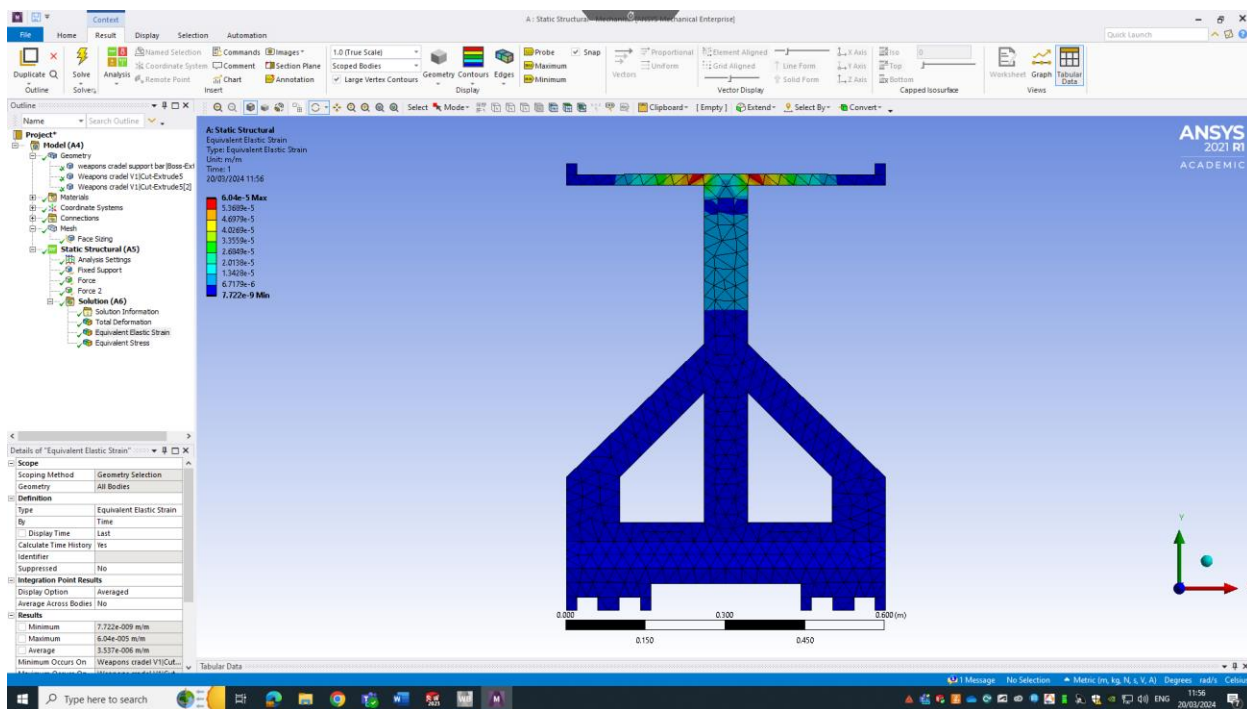
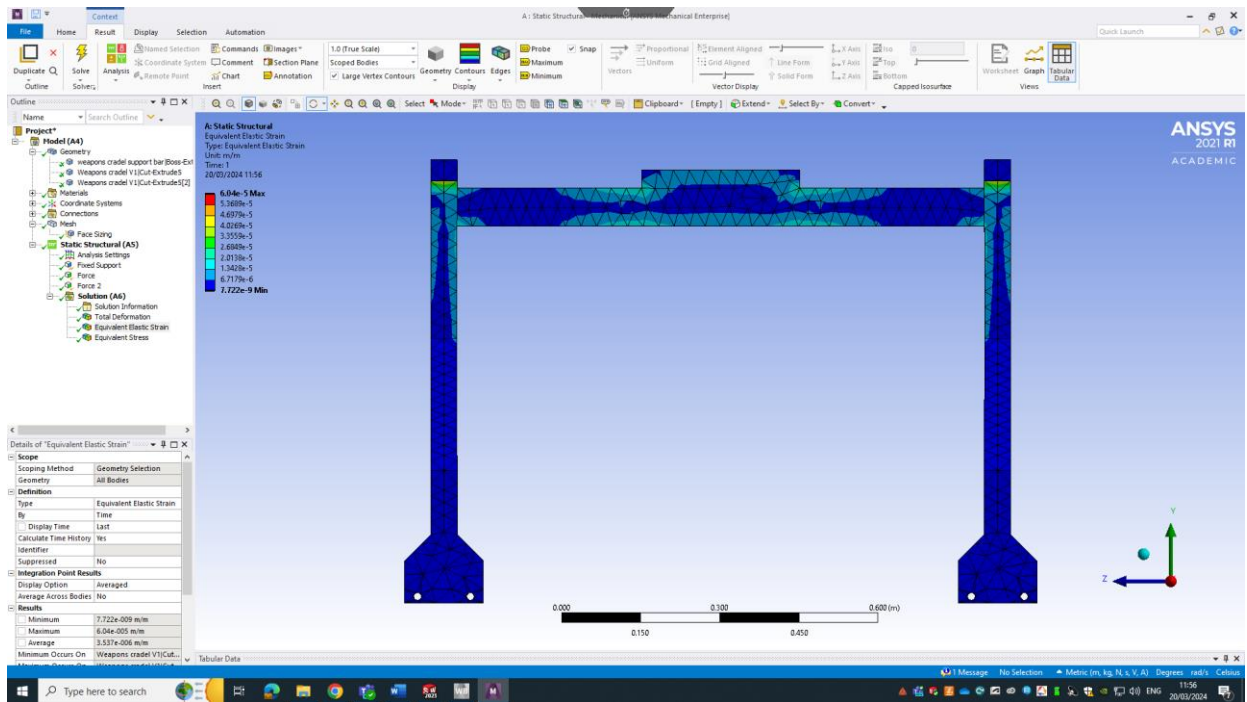


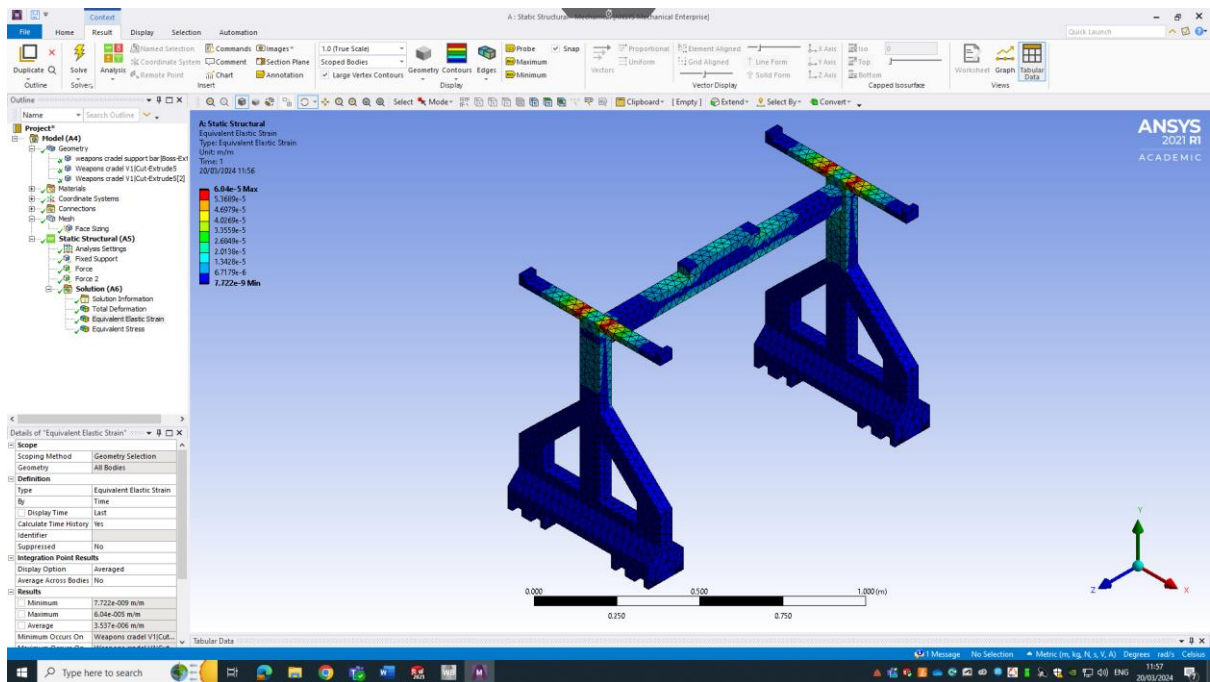
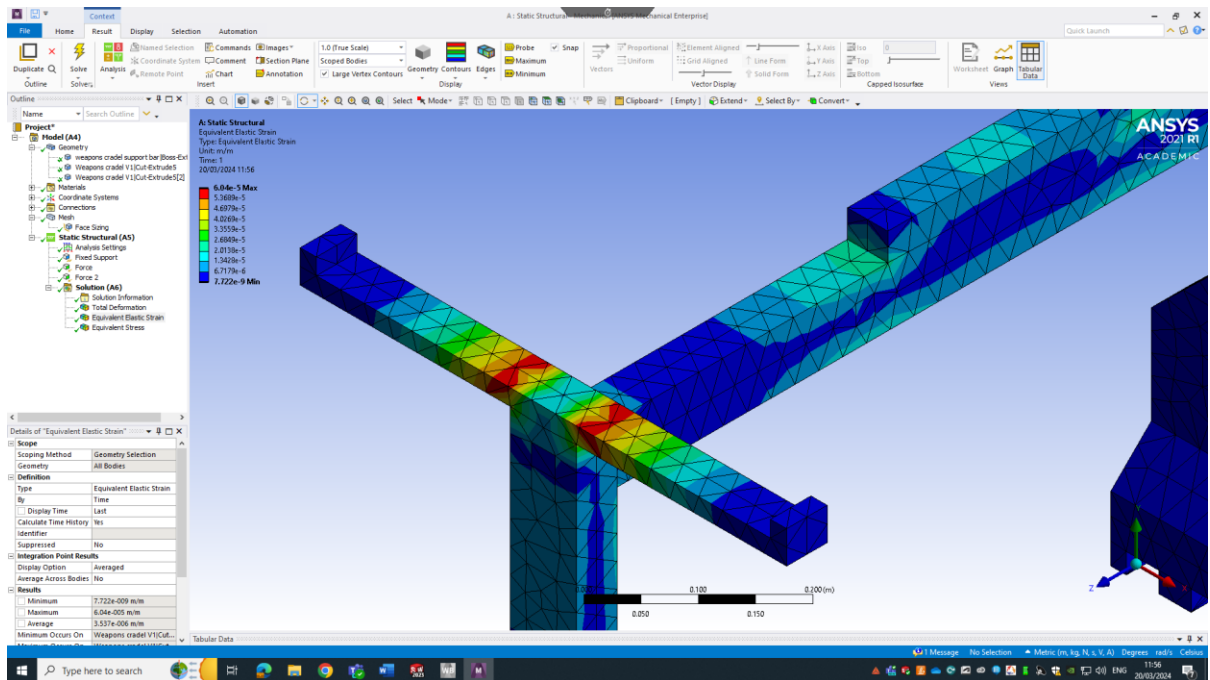




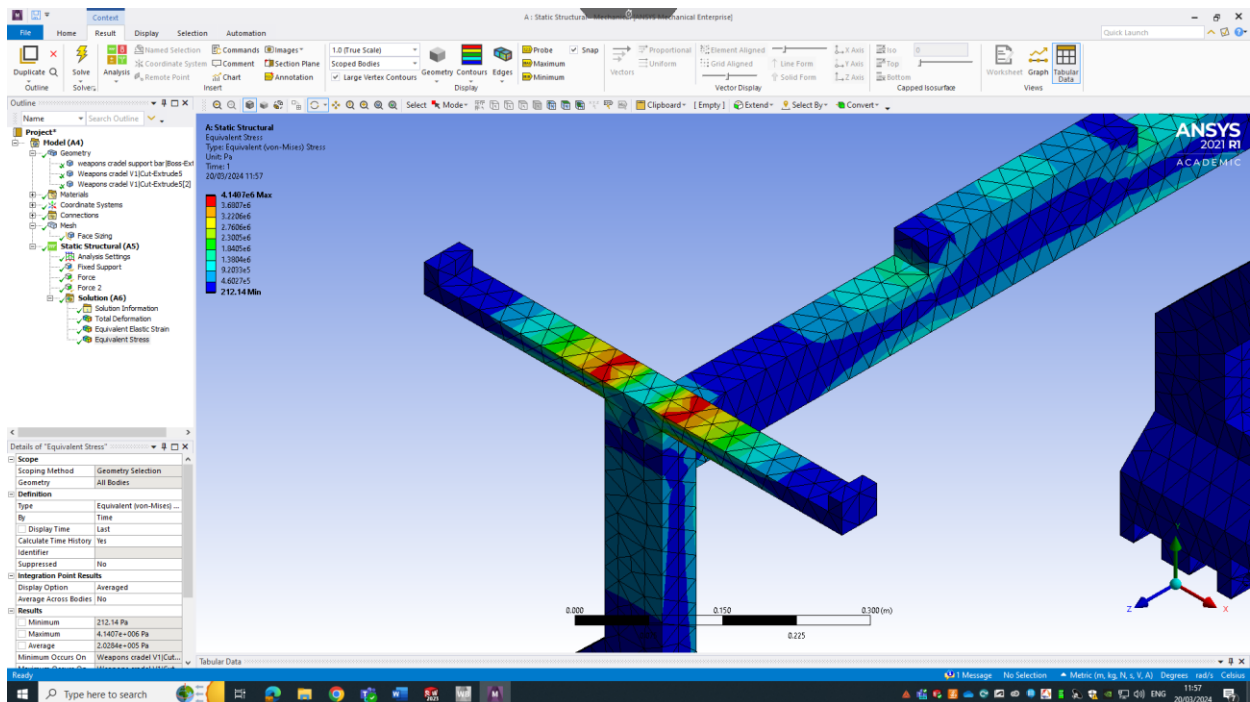
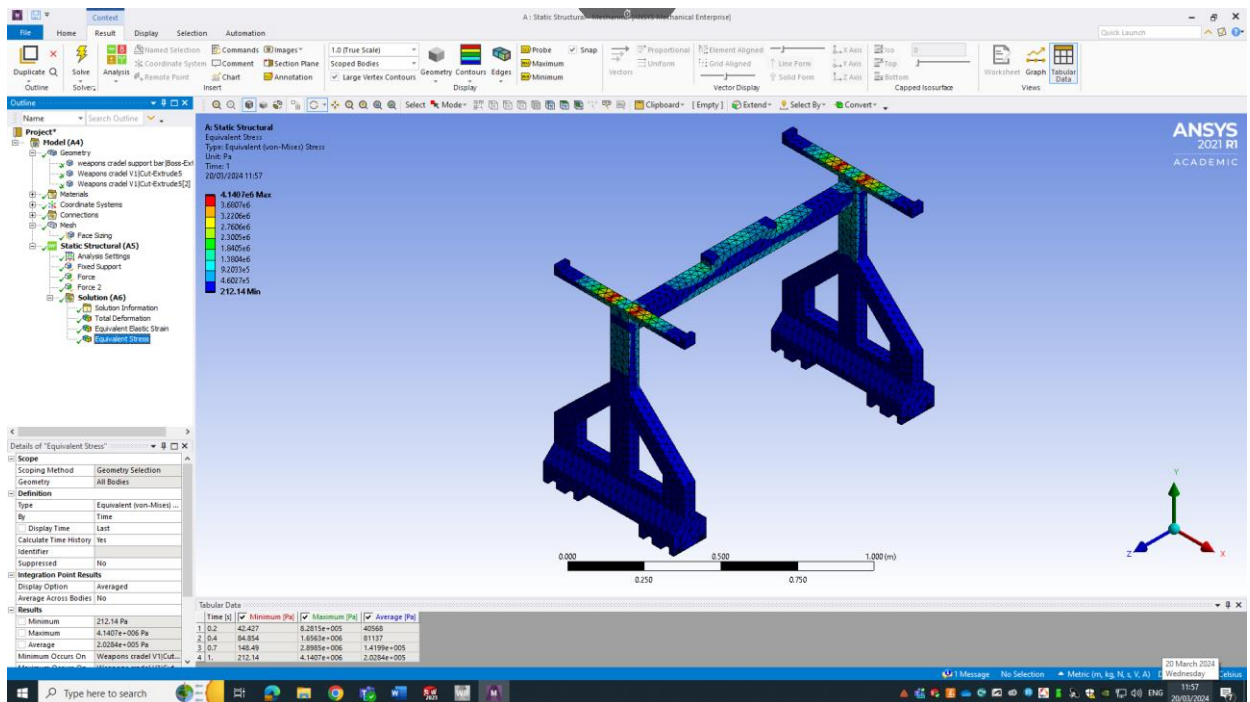
Showing Strain

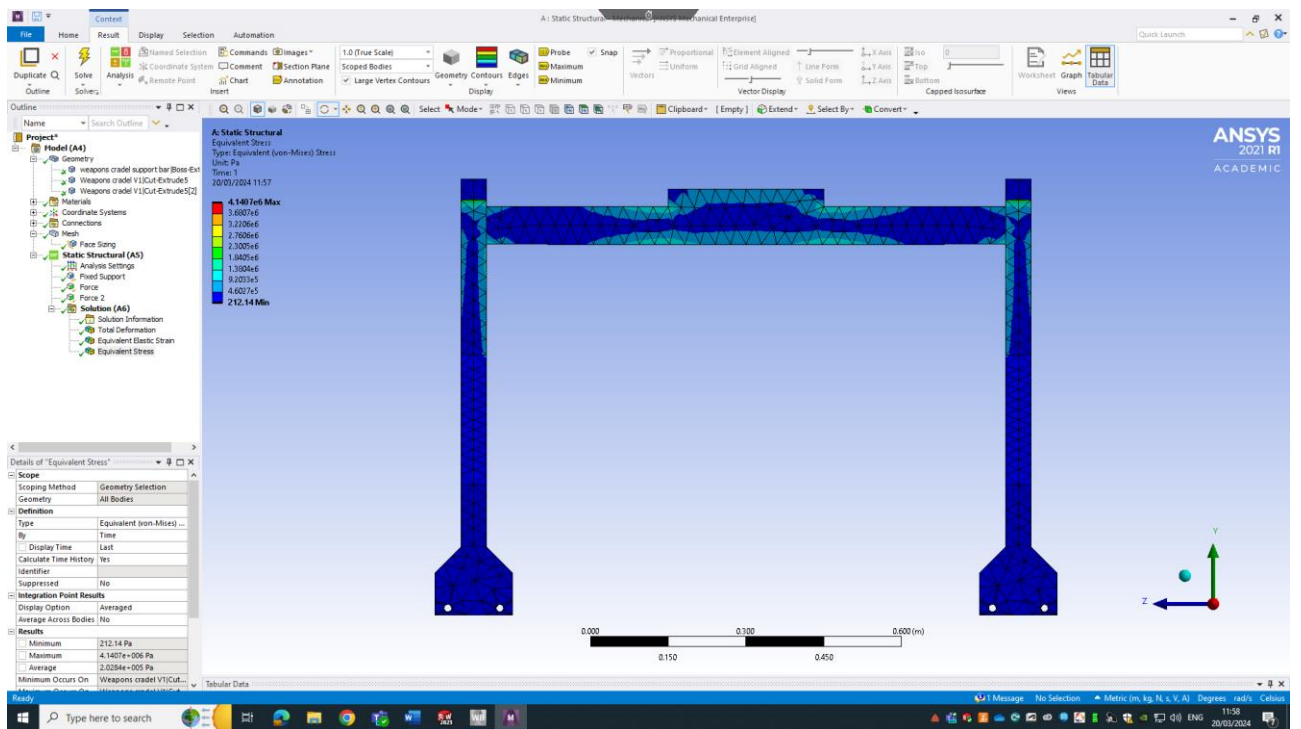
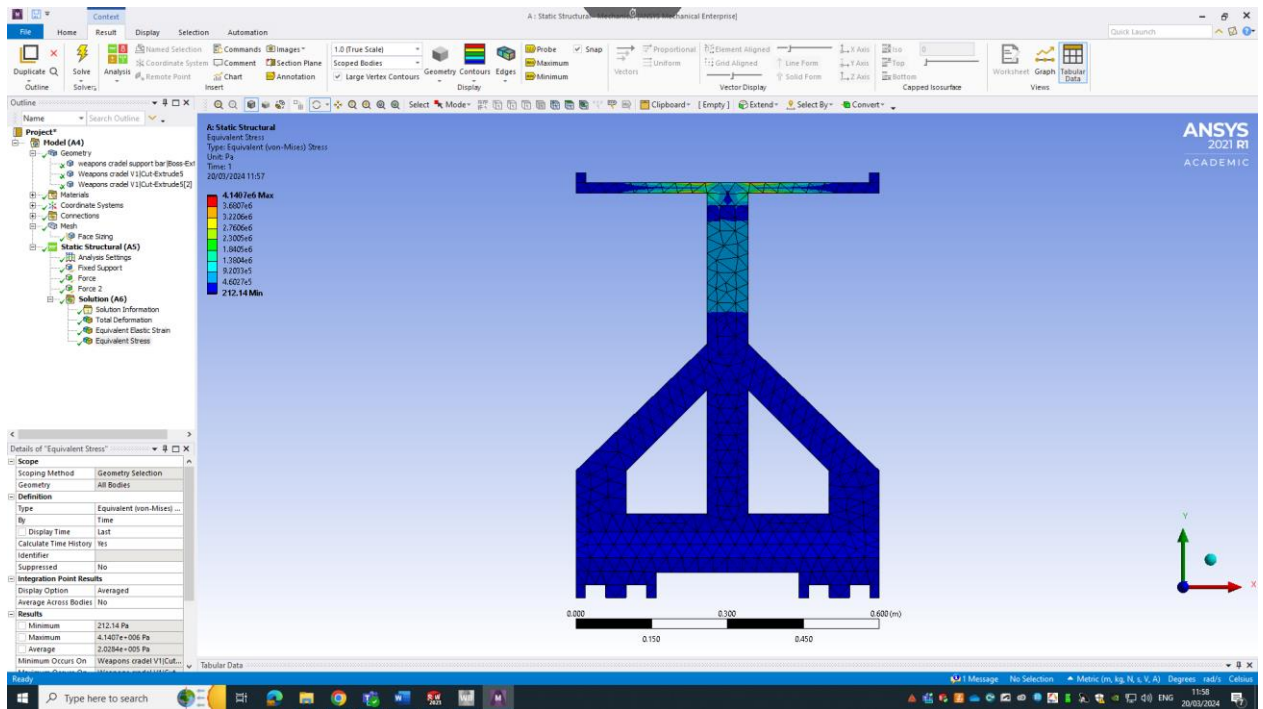






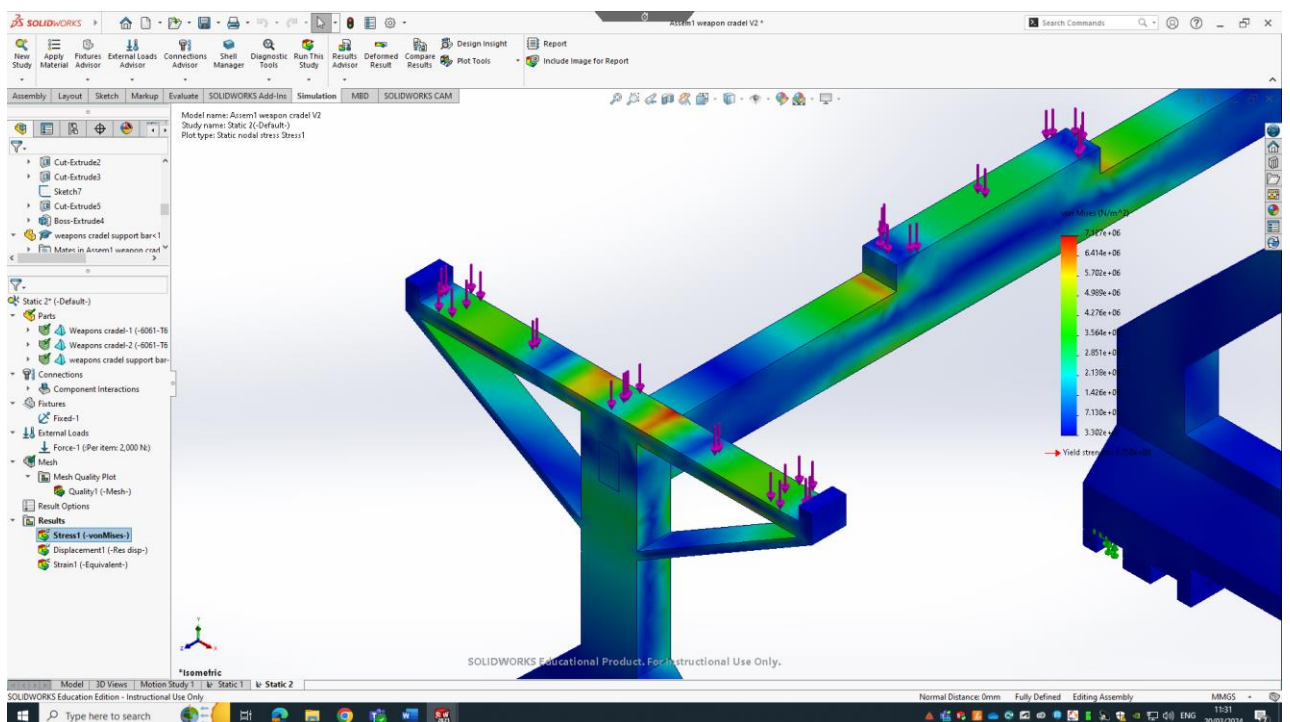
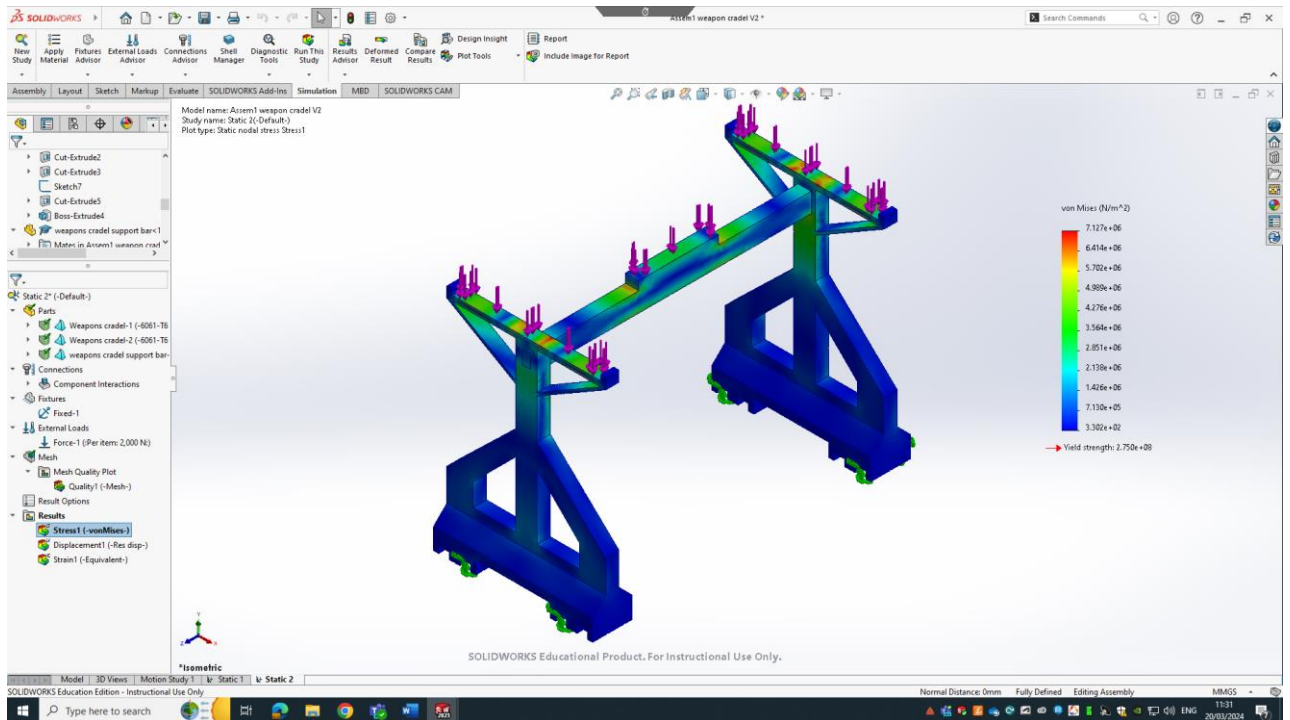
Showing Stress

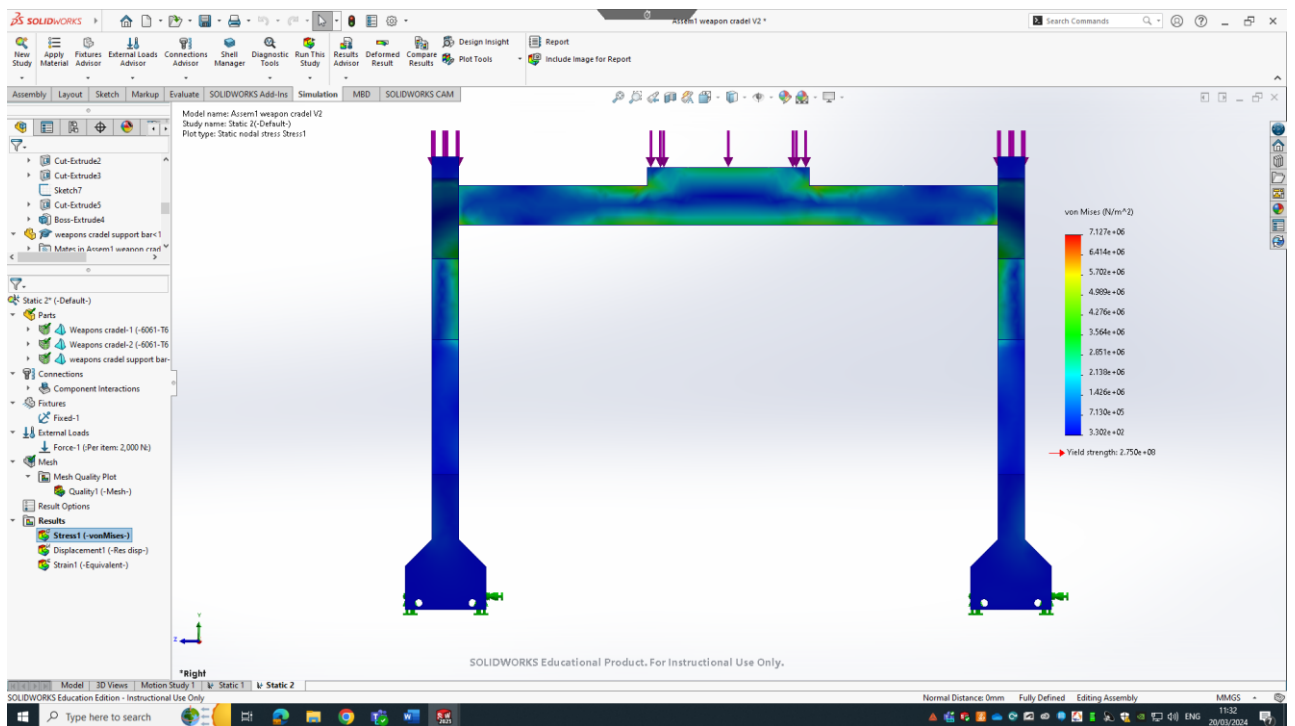
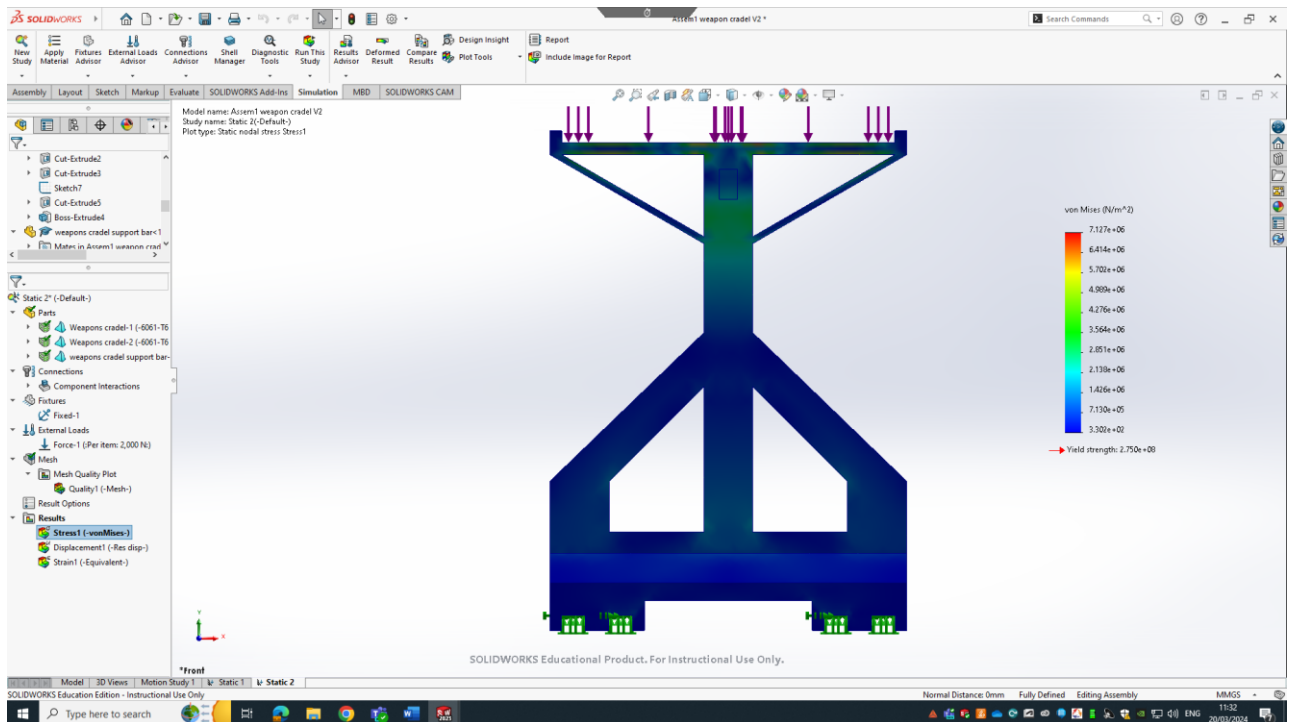




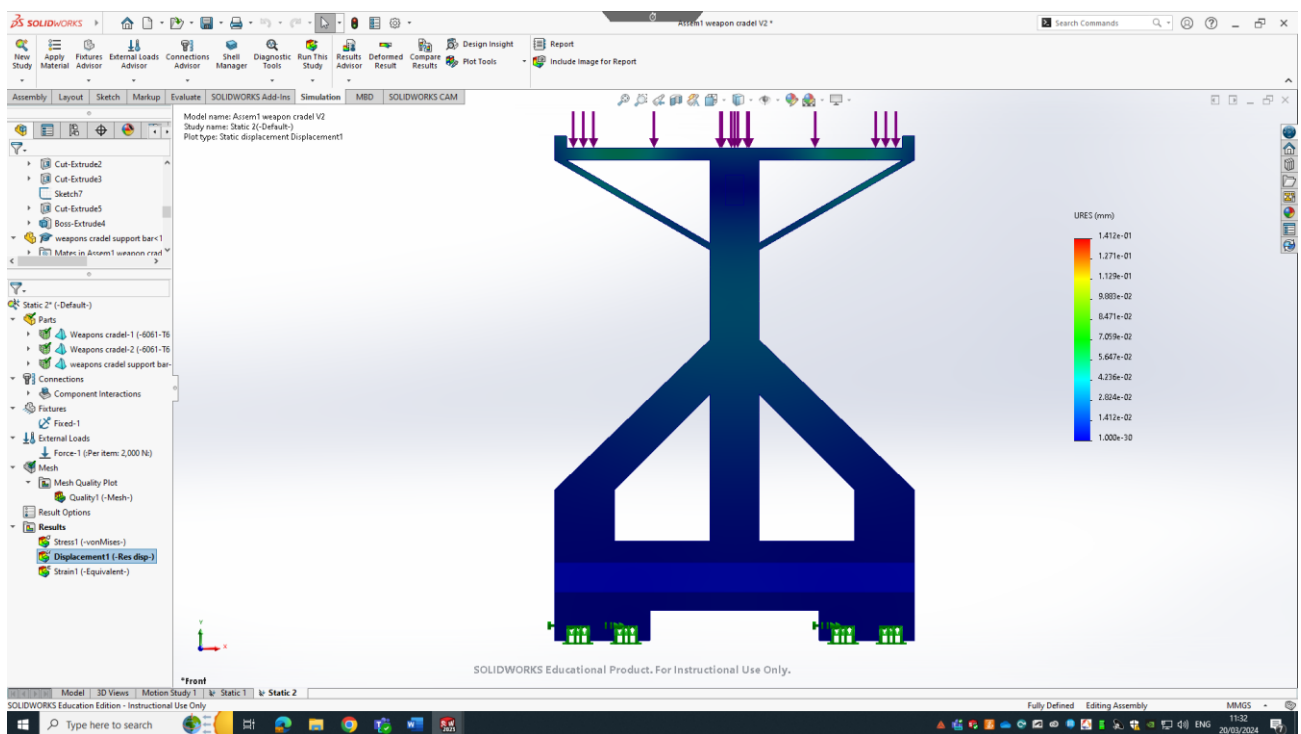
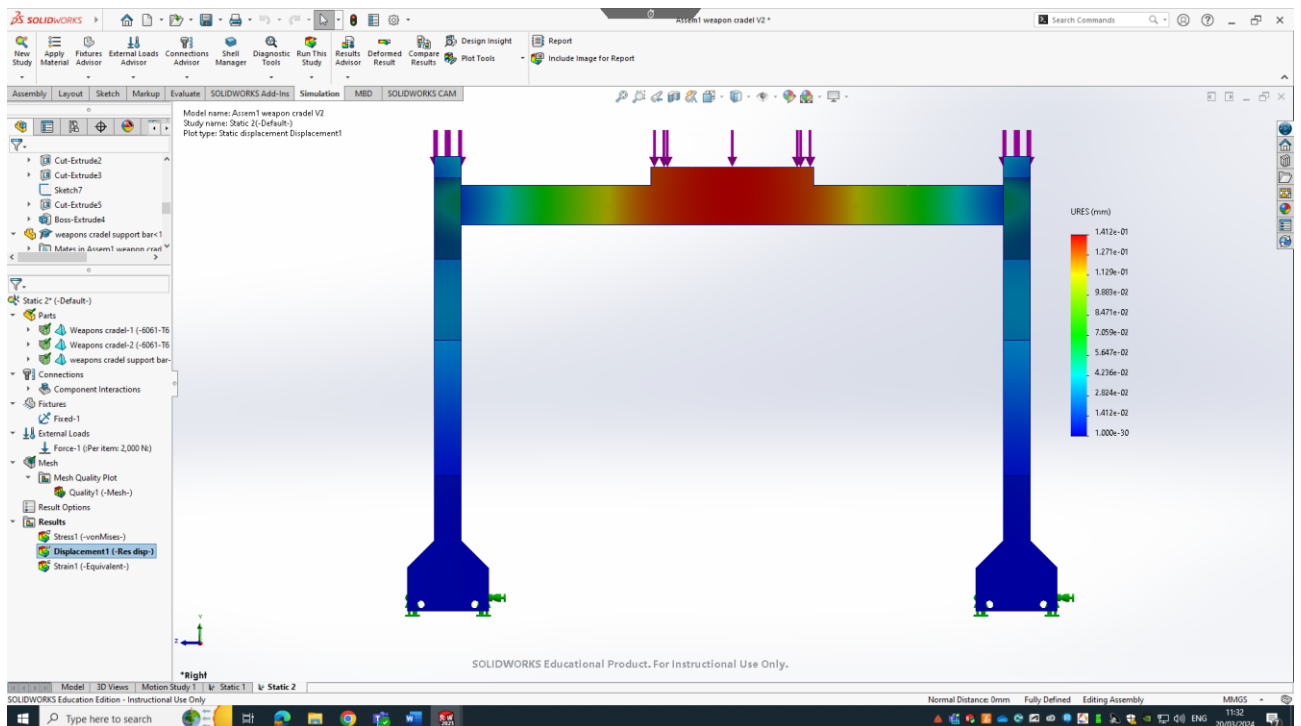
Assembly V2 simulation in solidworks representative load of 2000n across the three contact points, mesh size Max 30mm min 2.66202805mm

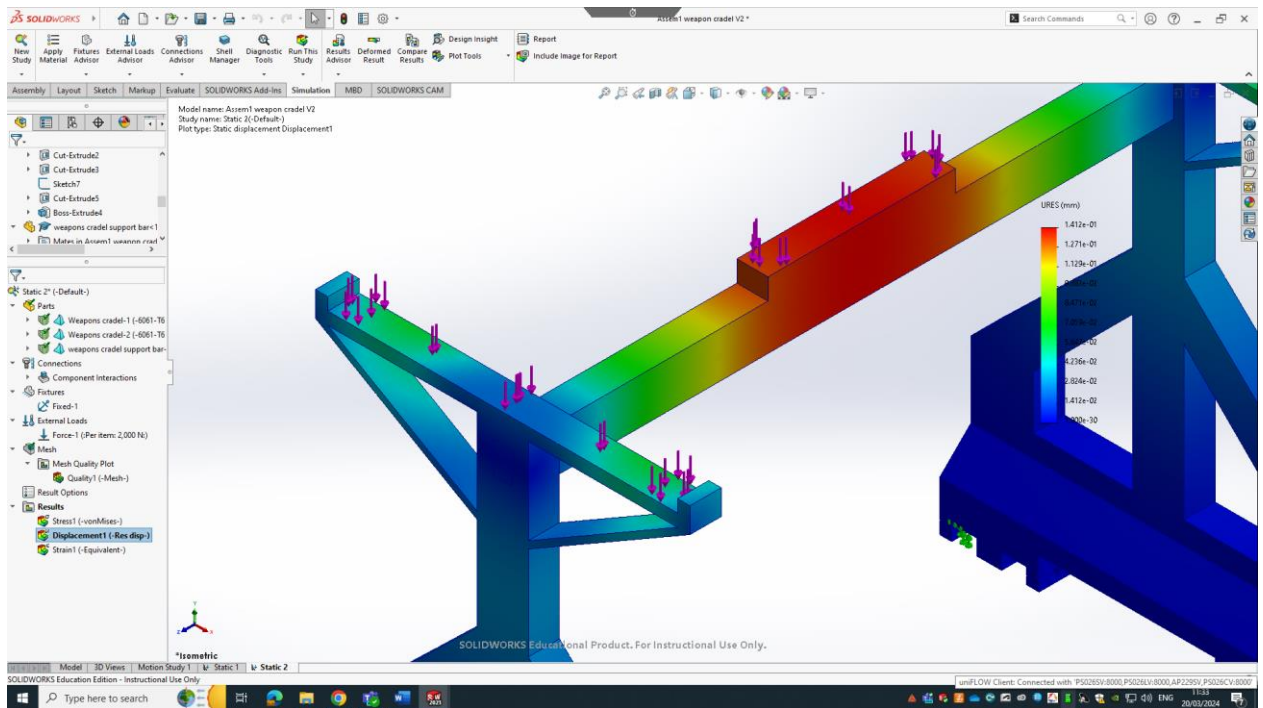
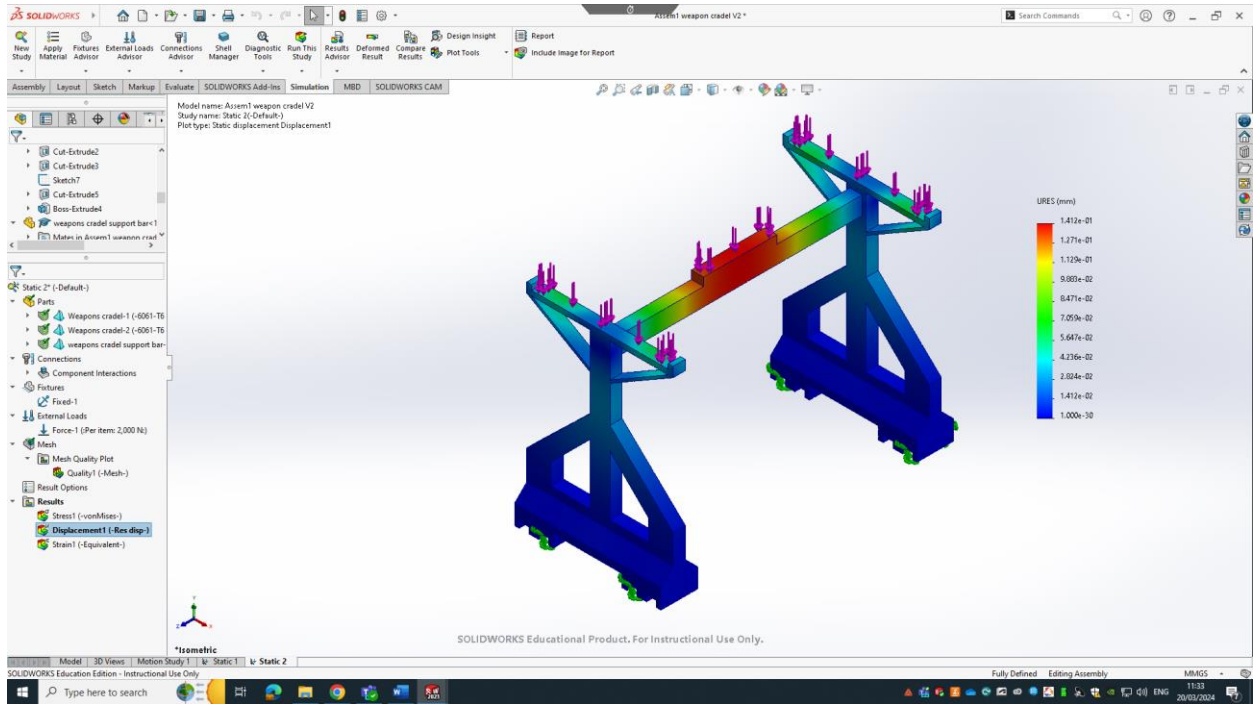
Showing Stress



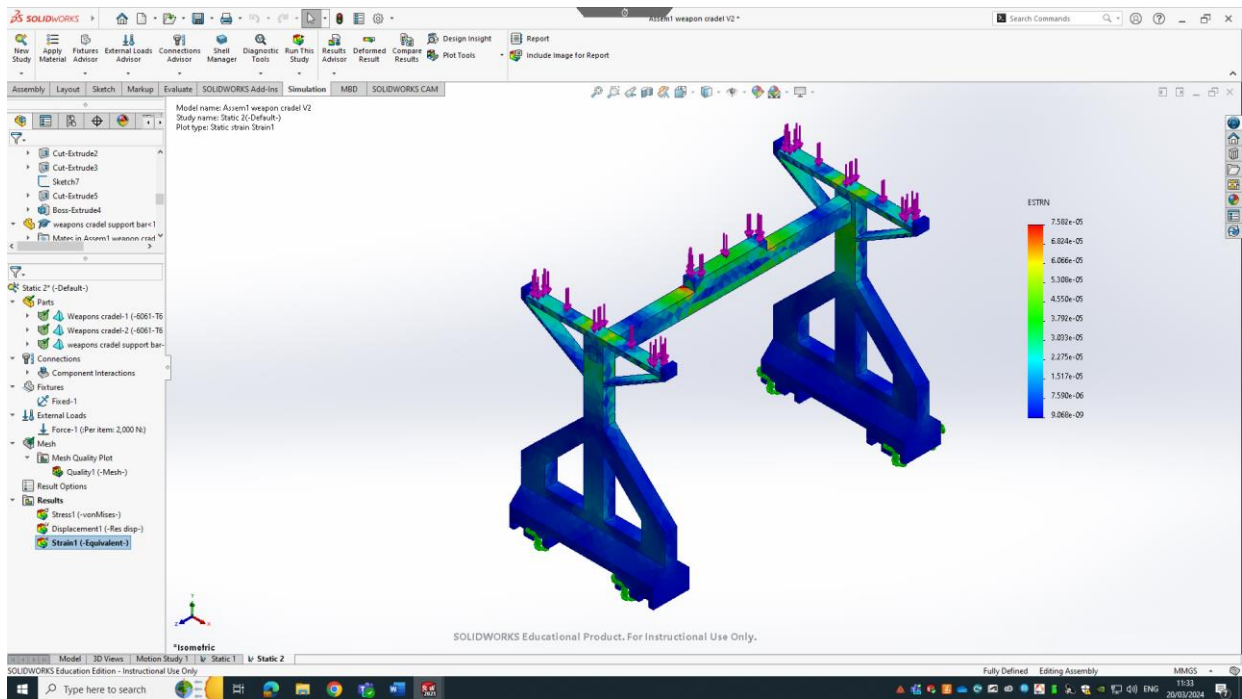
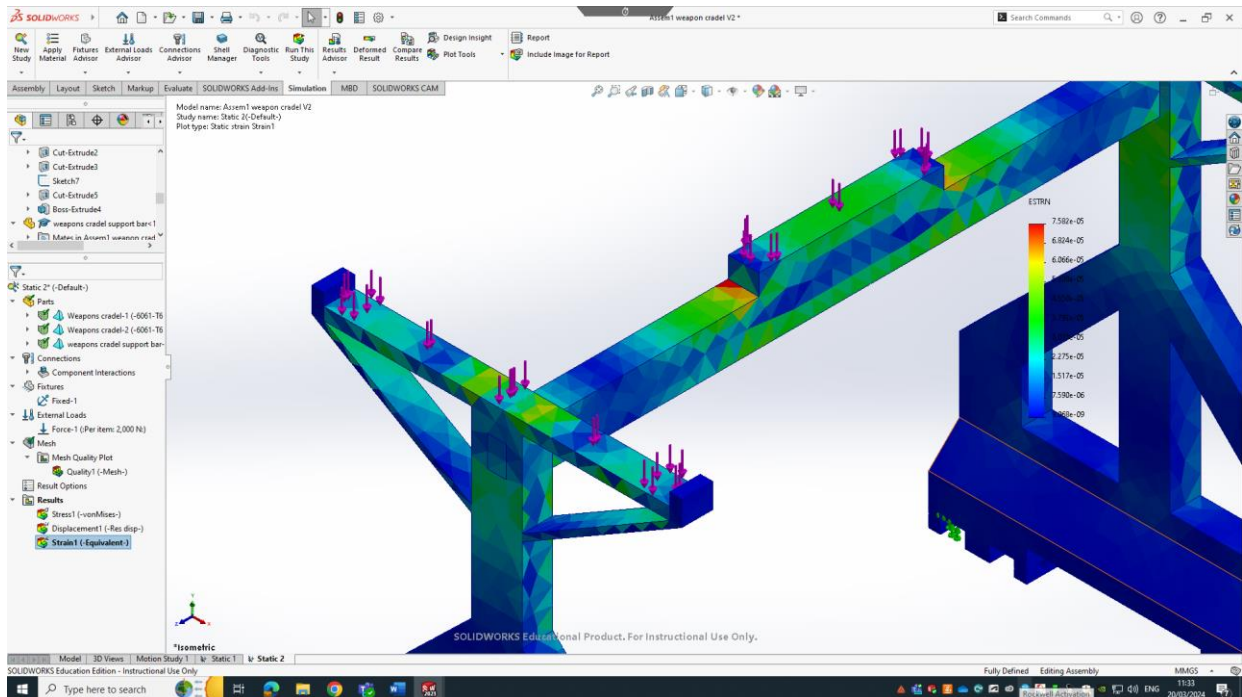


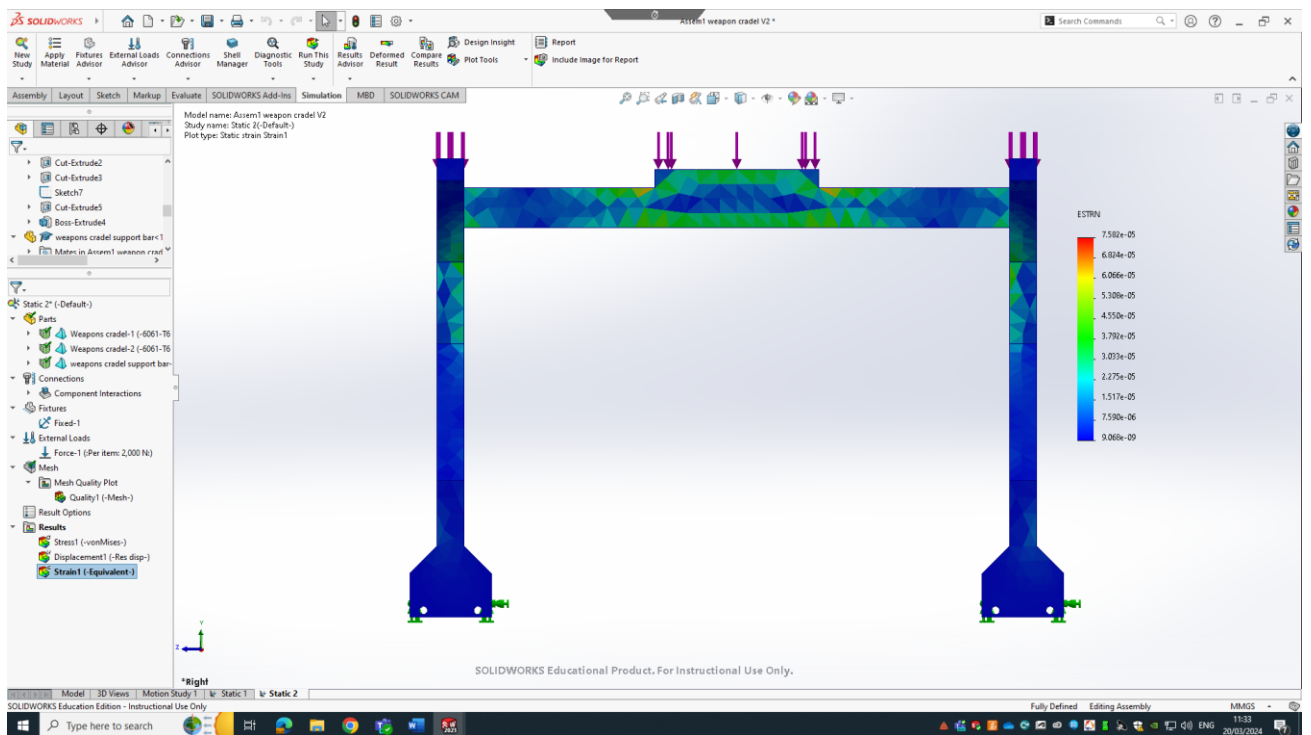
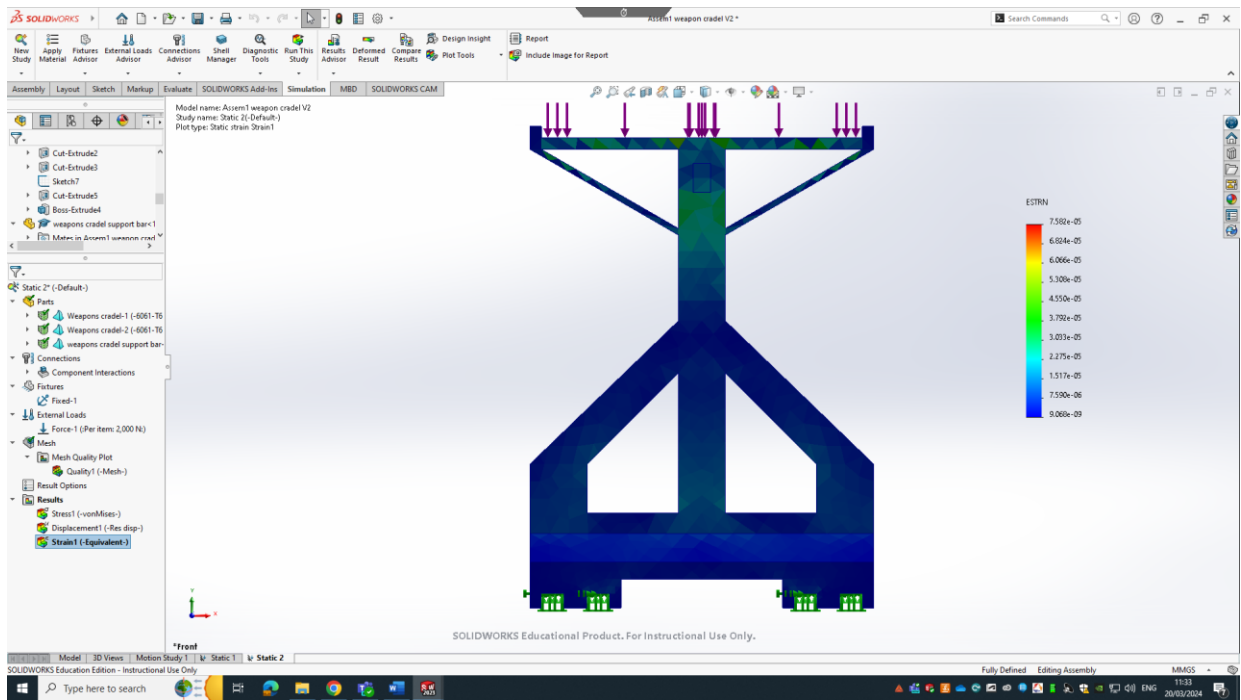
Showing Deformation





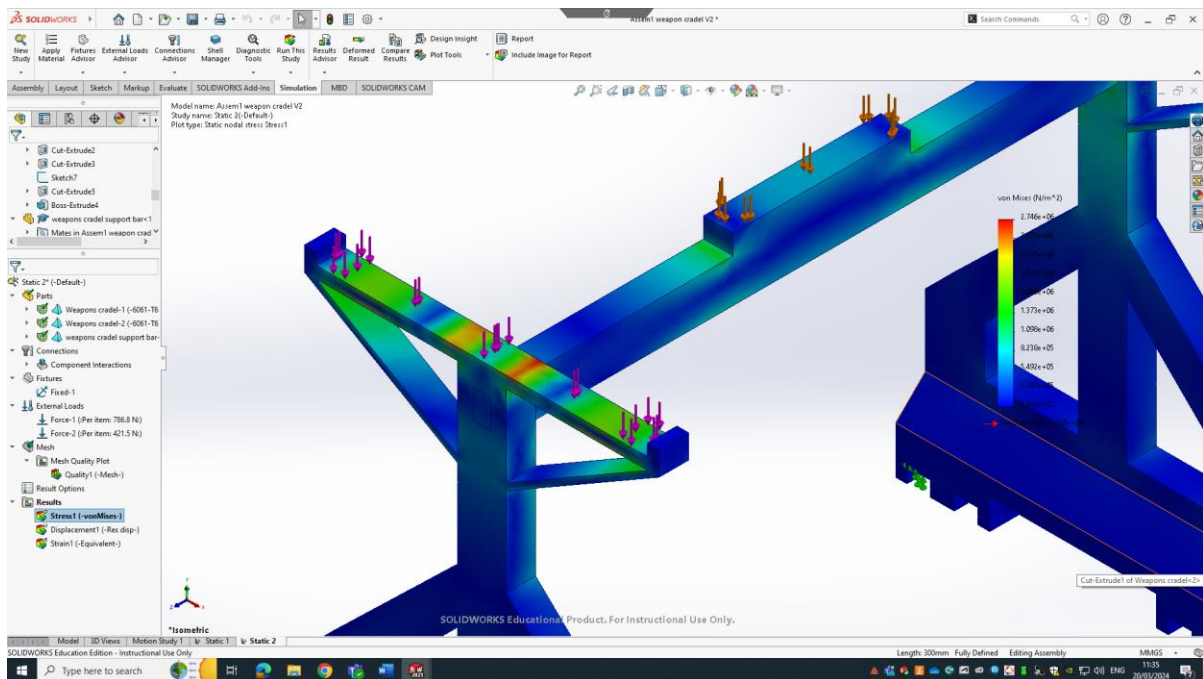
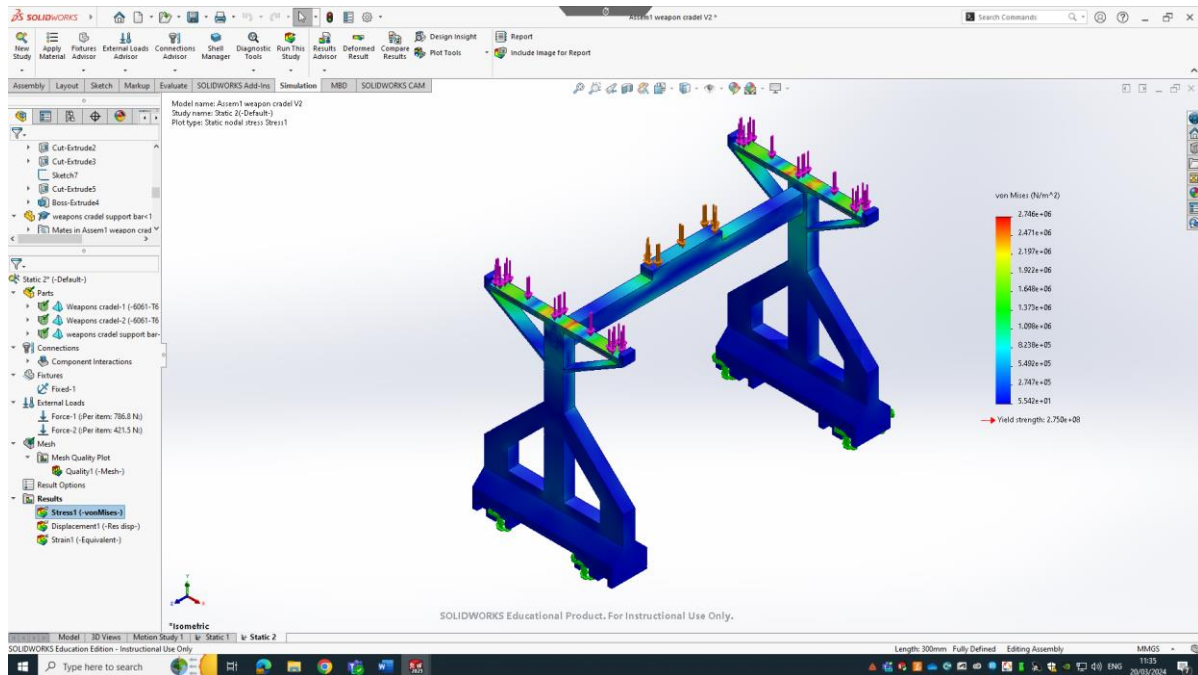
Showing Strain



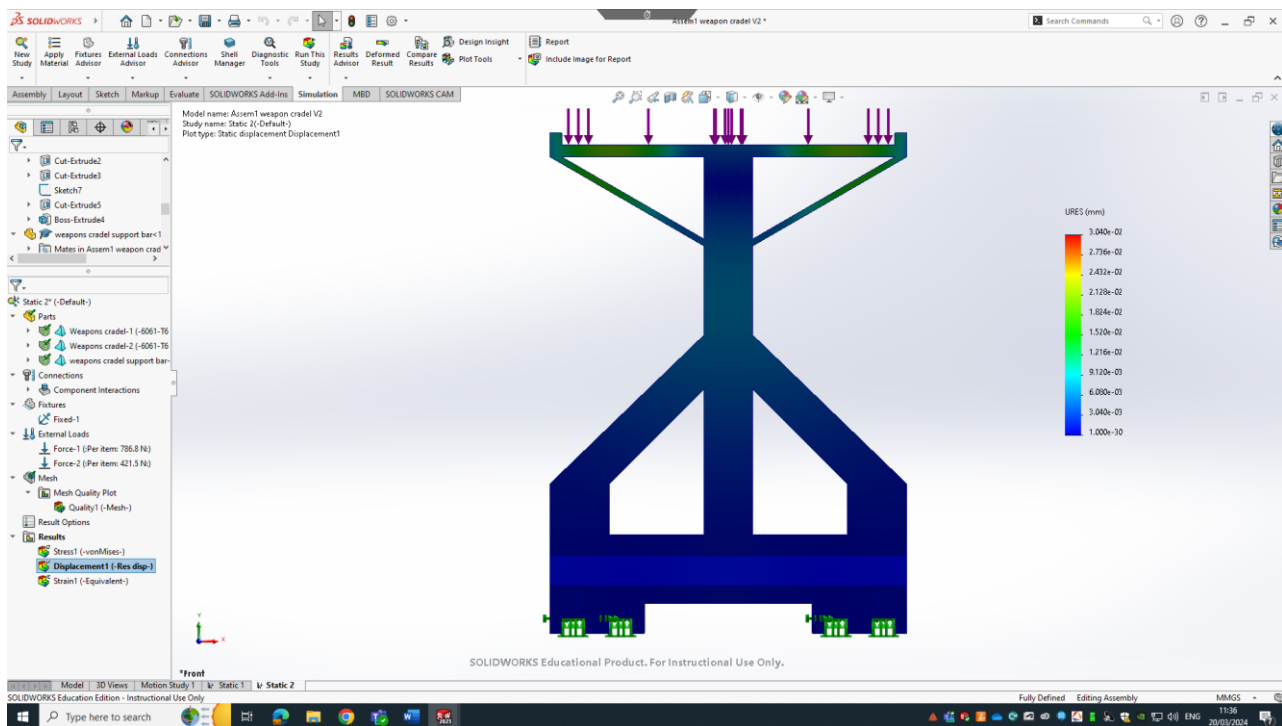
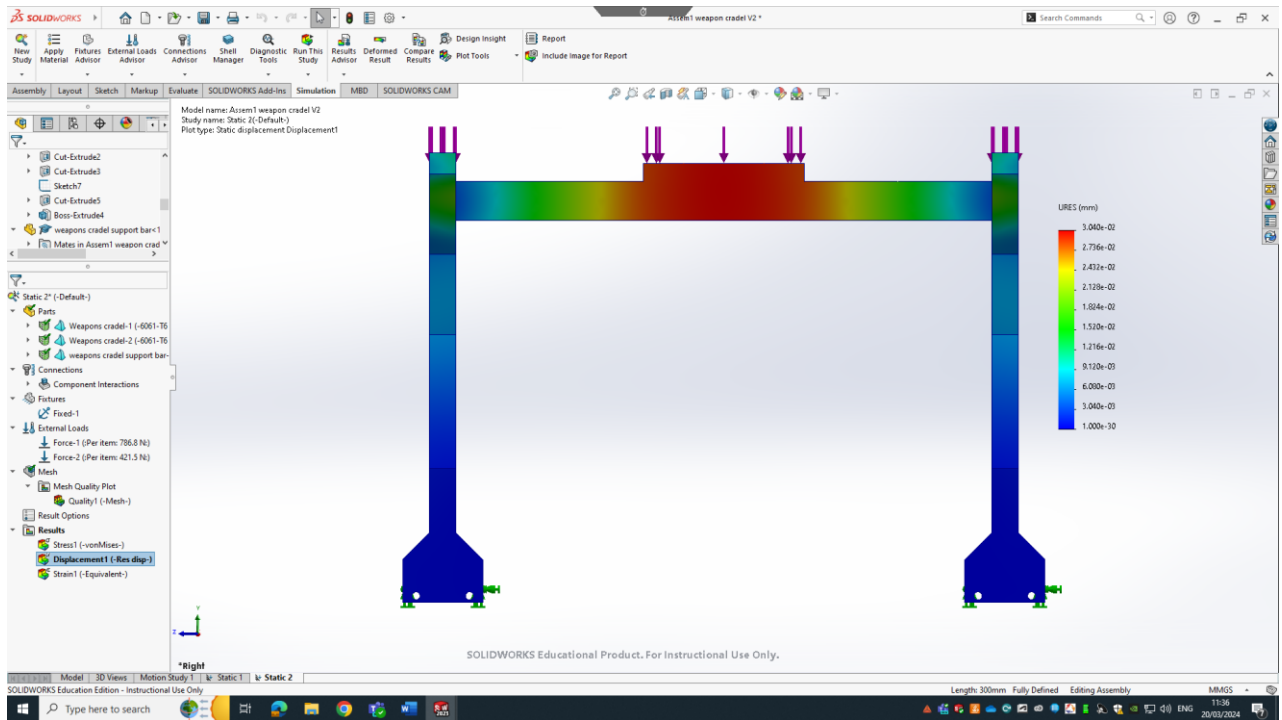


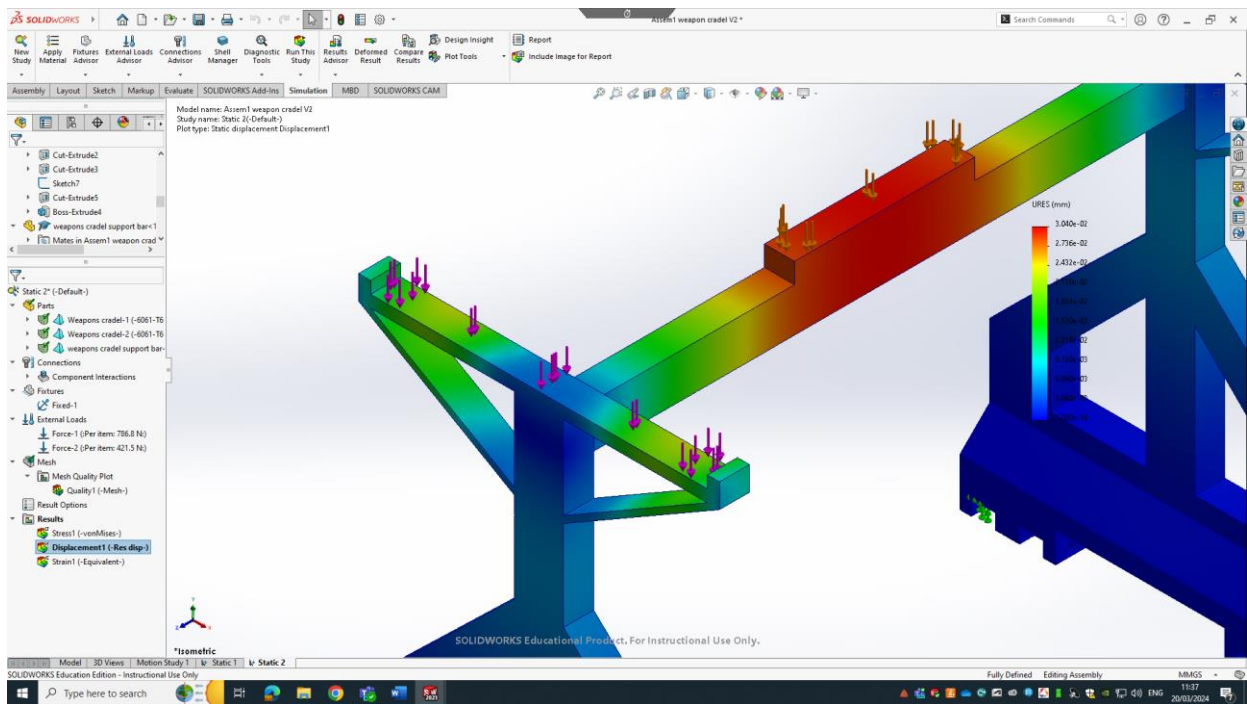
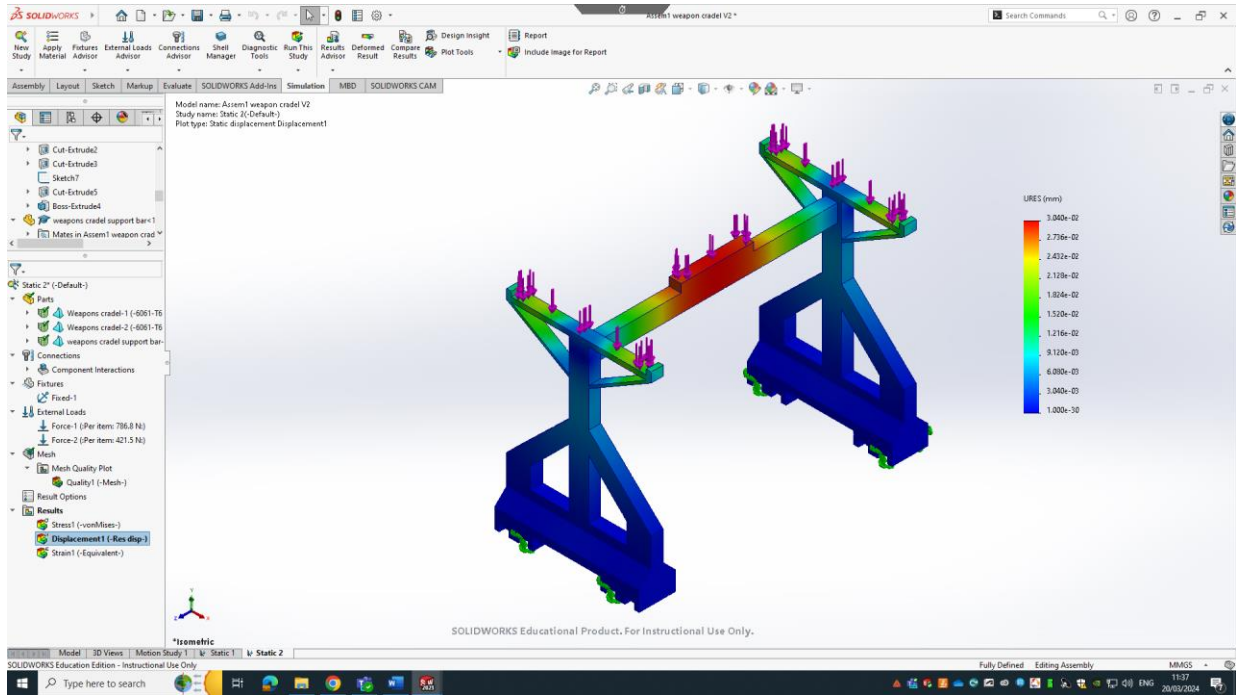
Assembly V2 simulation in solidworks representative load of 786.8n & 421.5n across the three contact points, mesh size Max 30mm min 2.66202805mm

Showing Stress

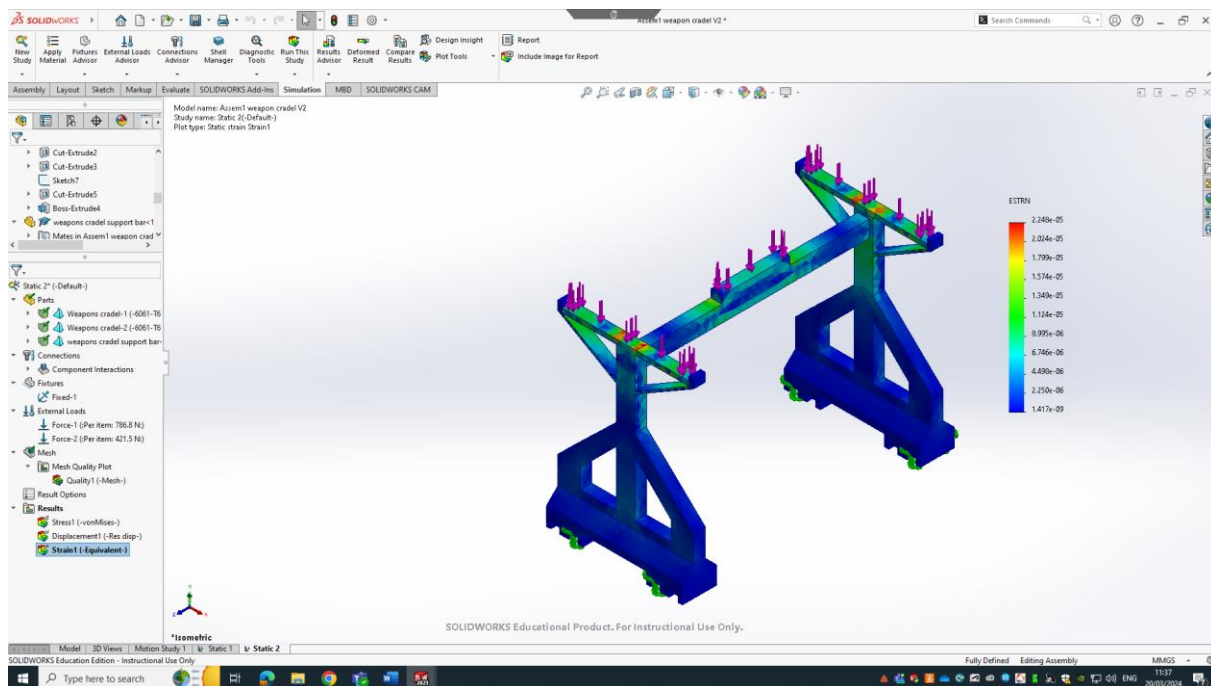
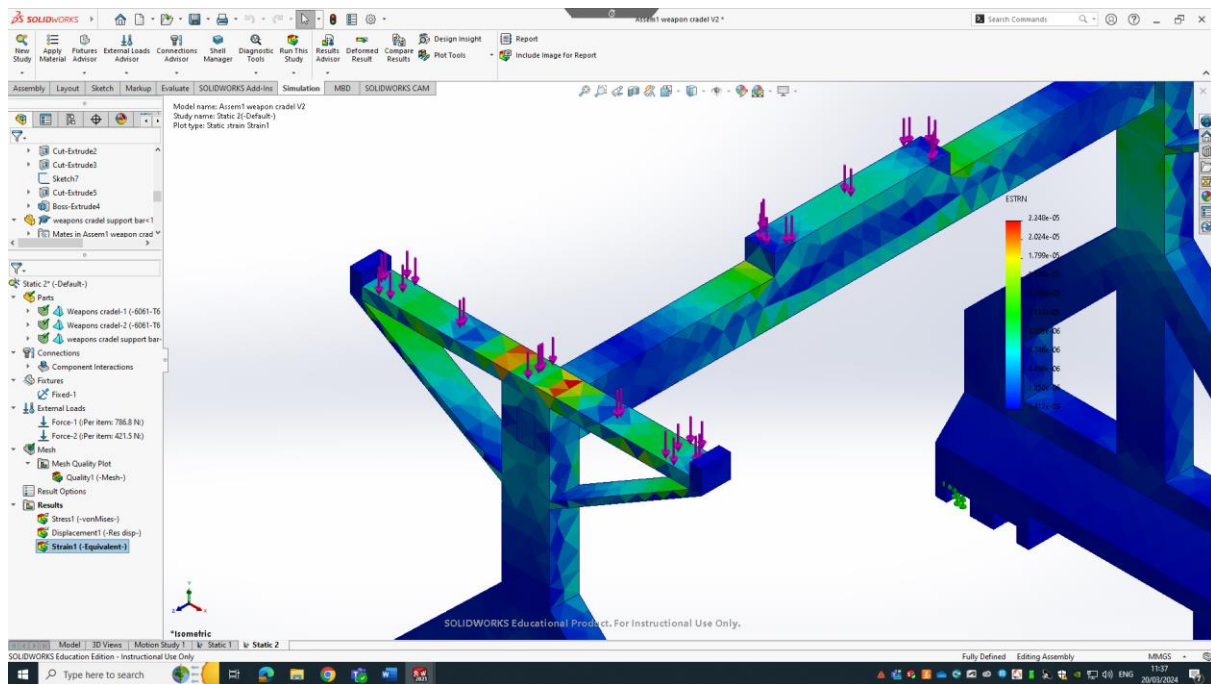


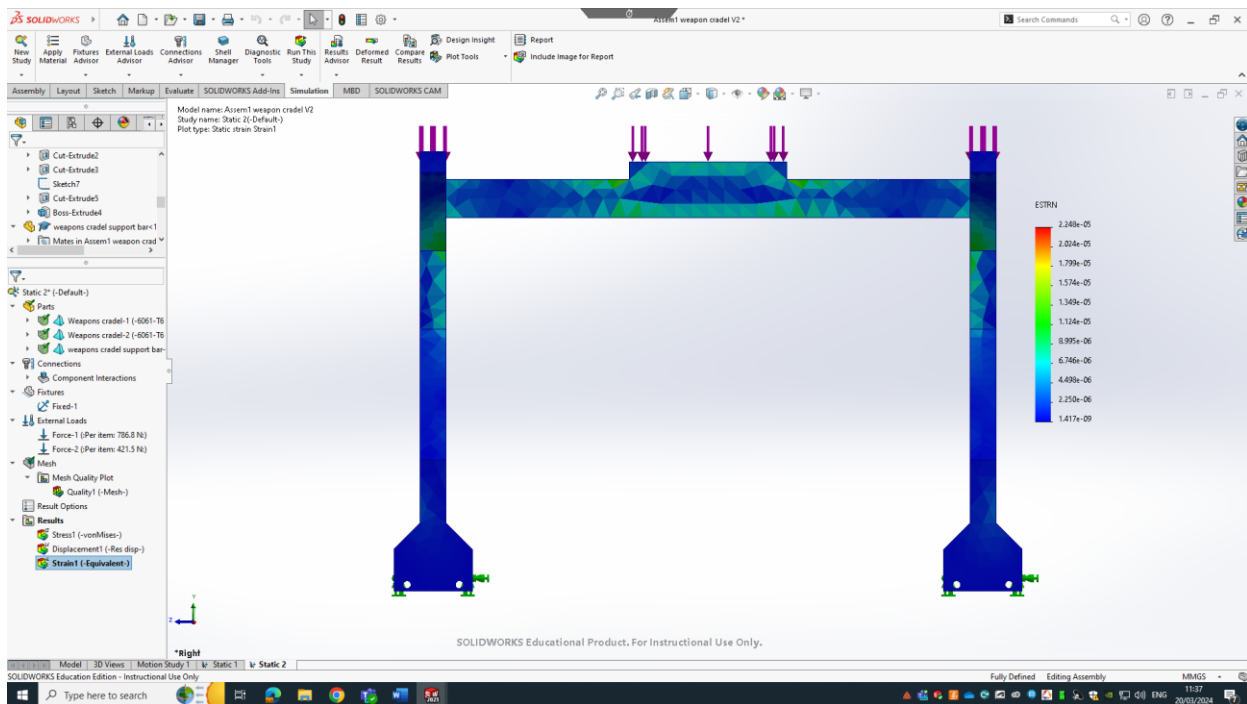
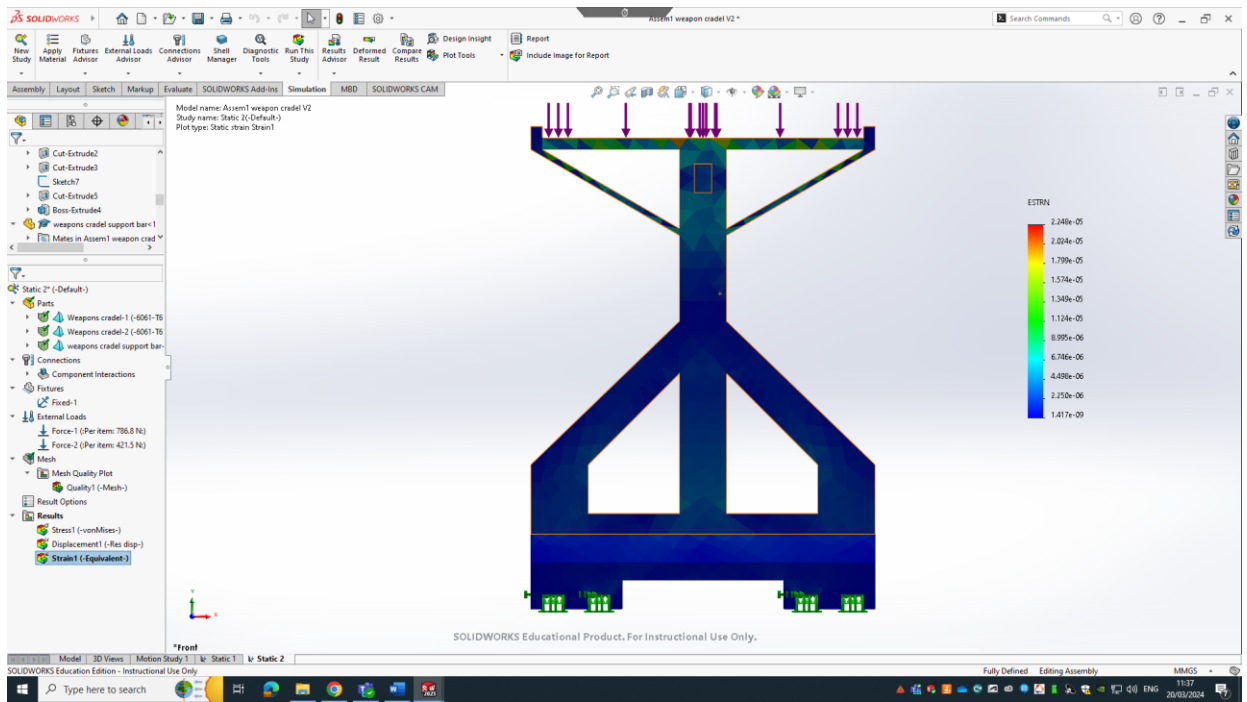
Showing Deformation





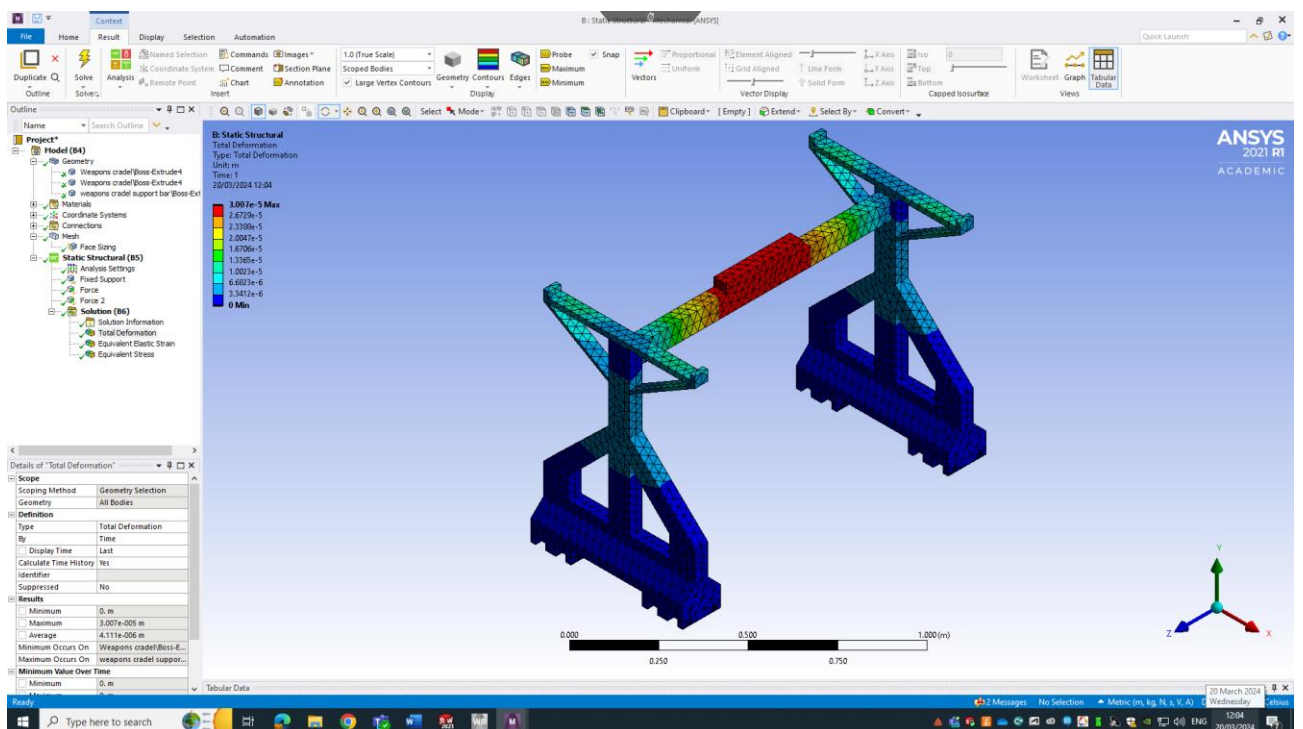
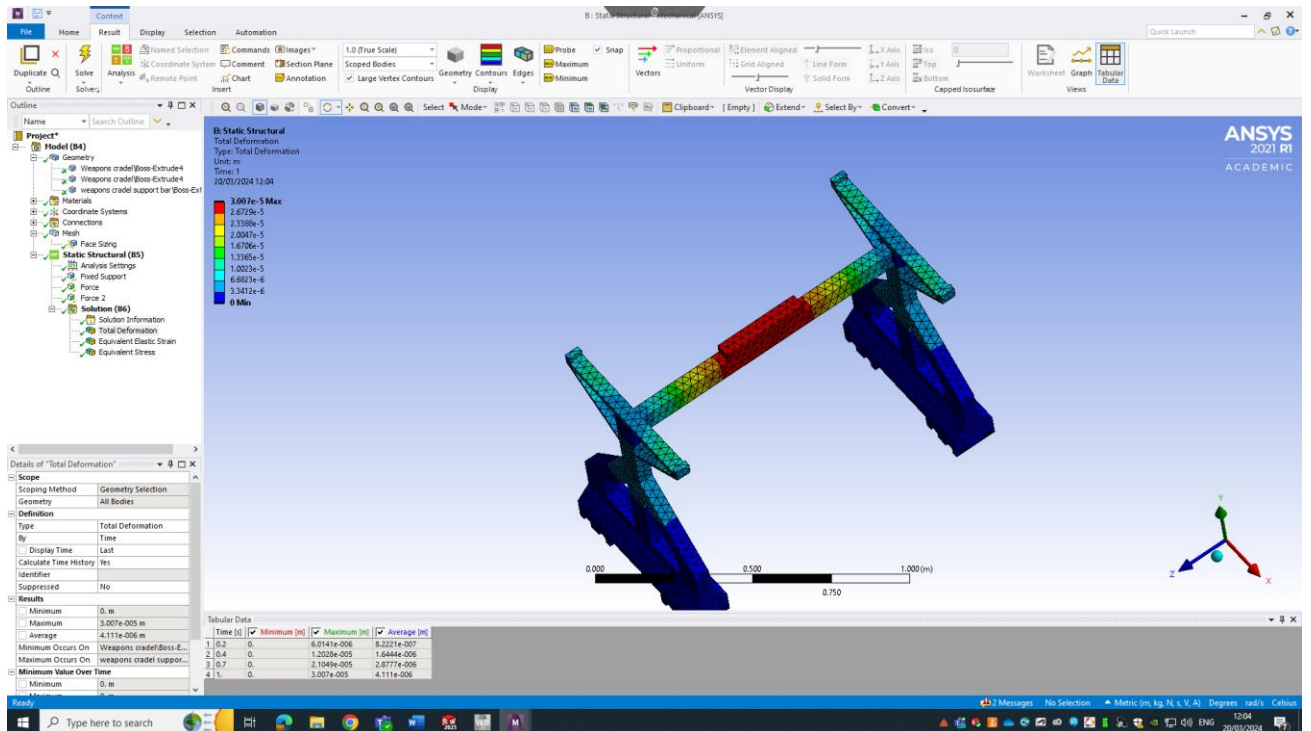
Showing Strain

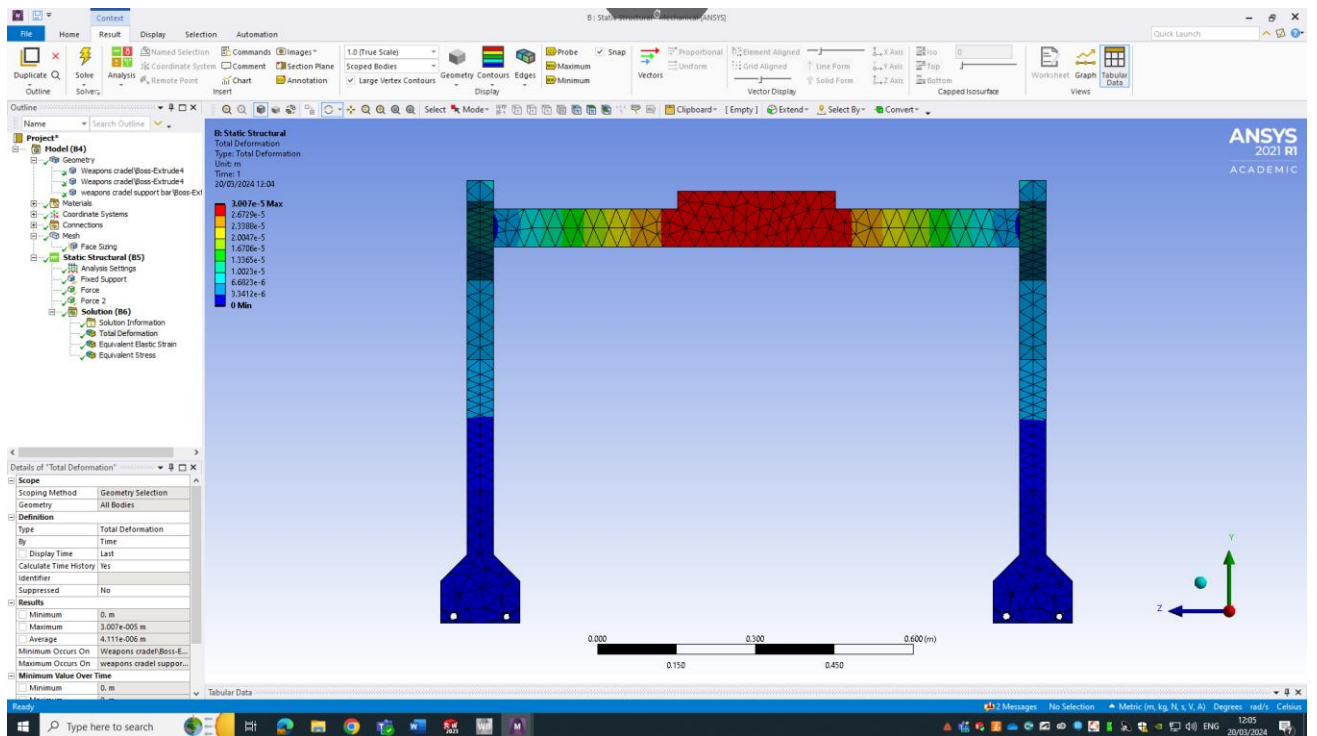
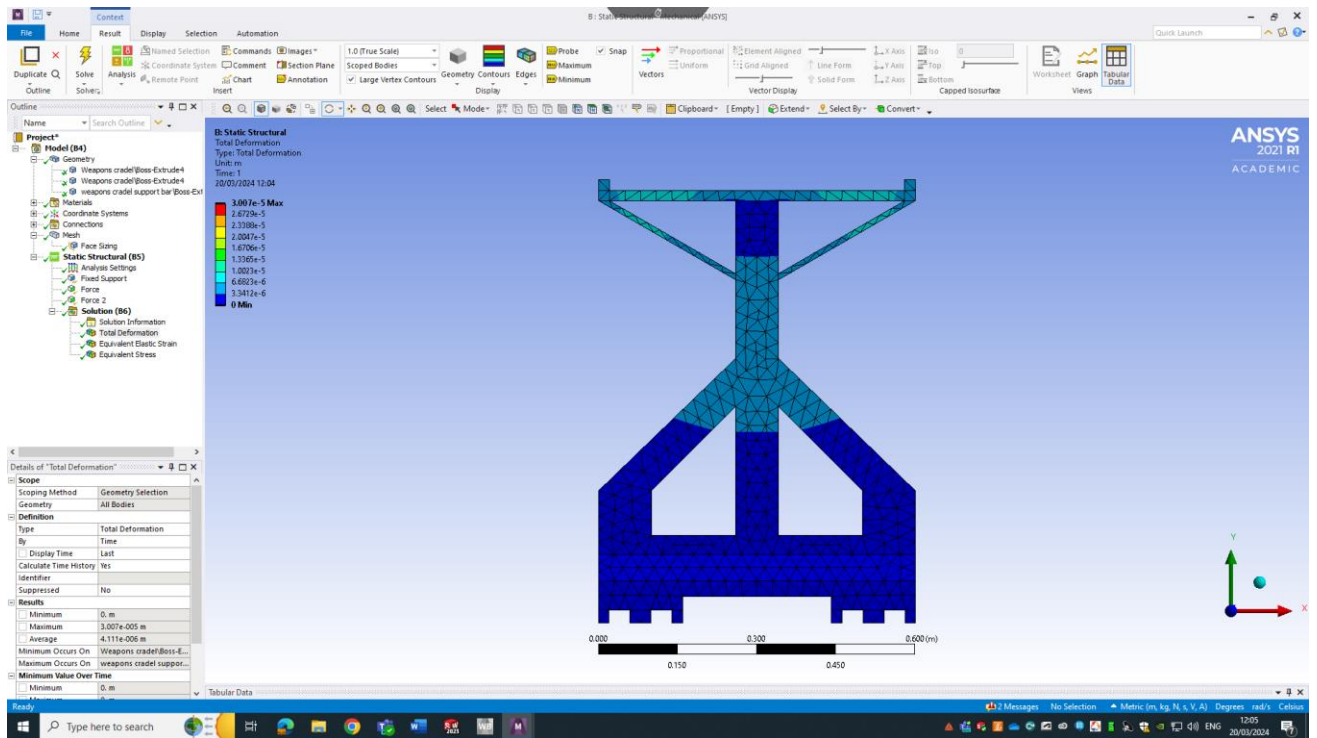


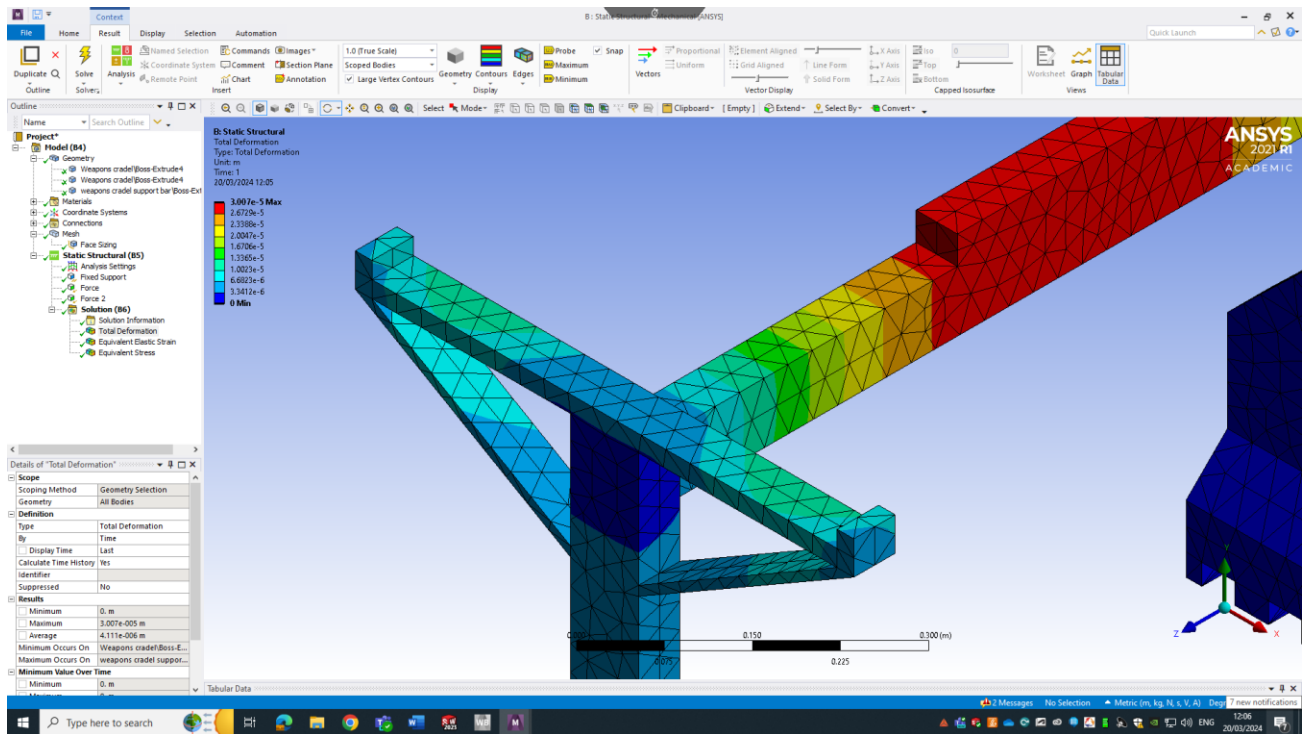


Assembly V2 simulation in Ansys representative load of 786.8n & 421.5n across the three contact points, mesh size 30mm

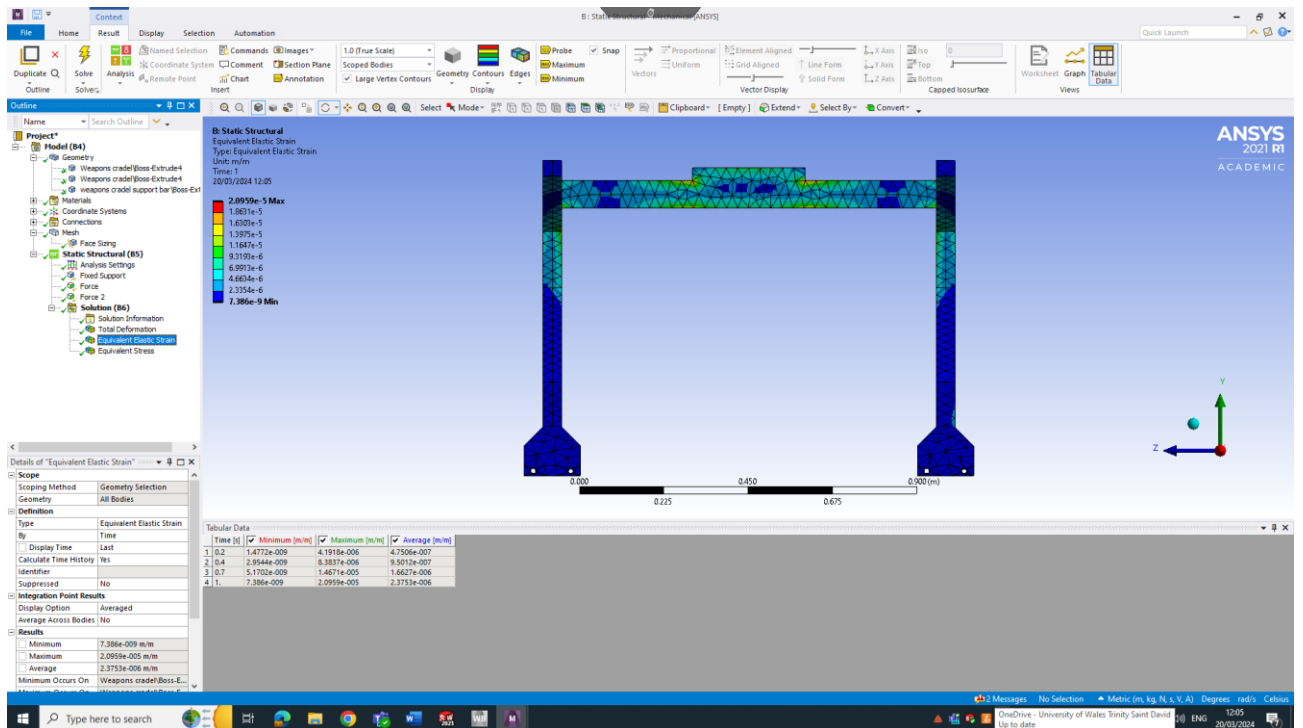
Showing Deformation

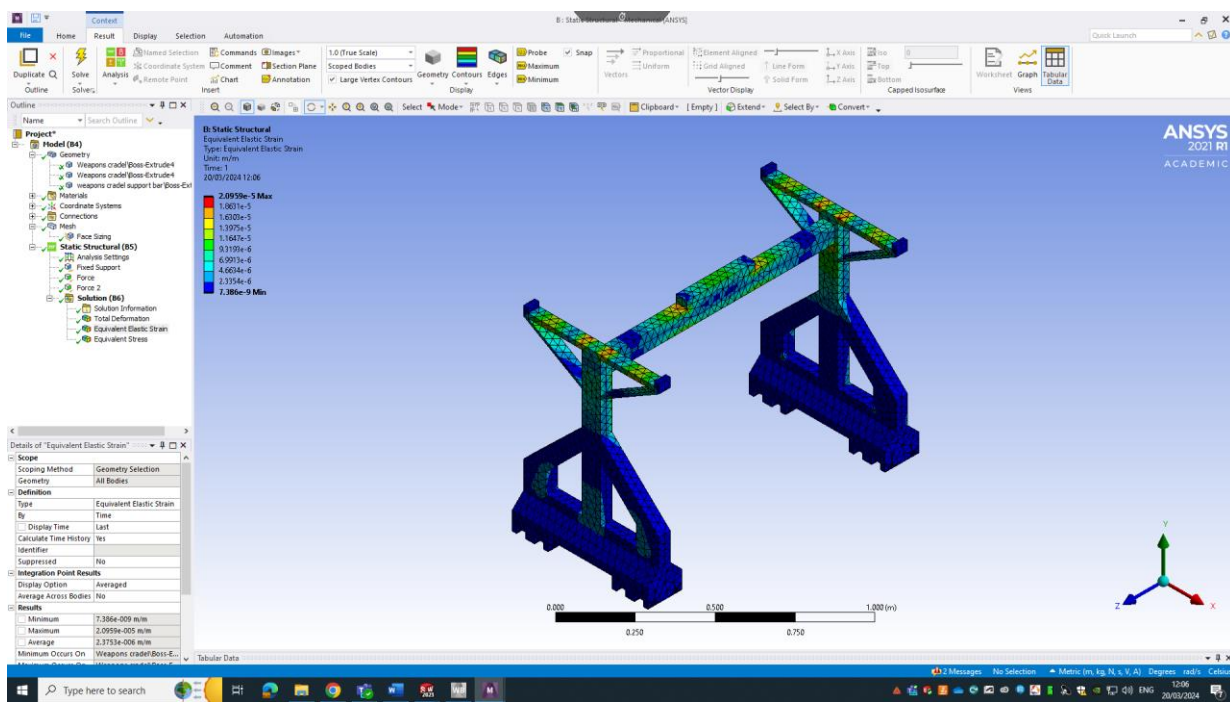
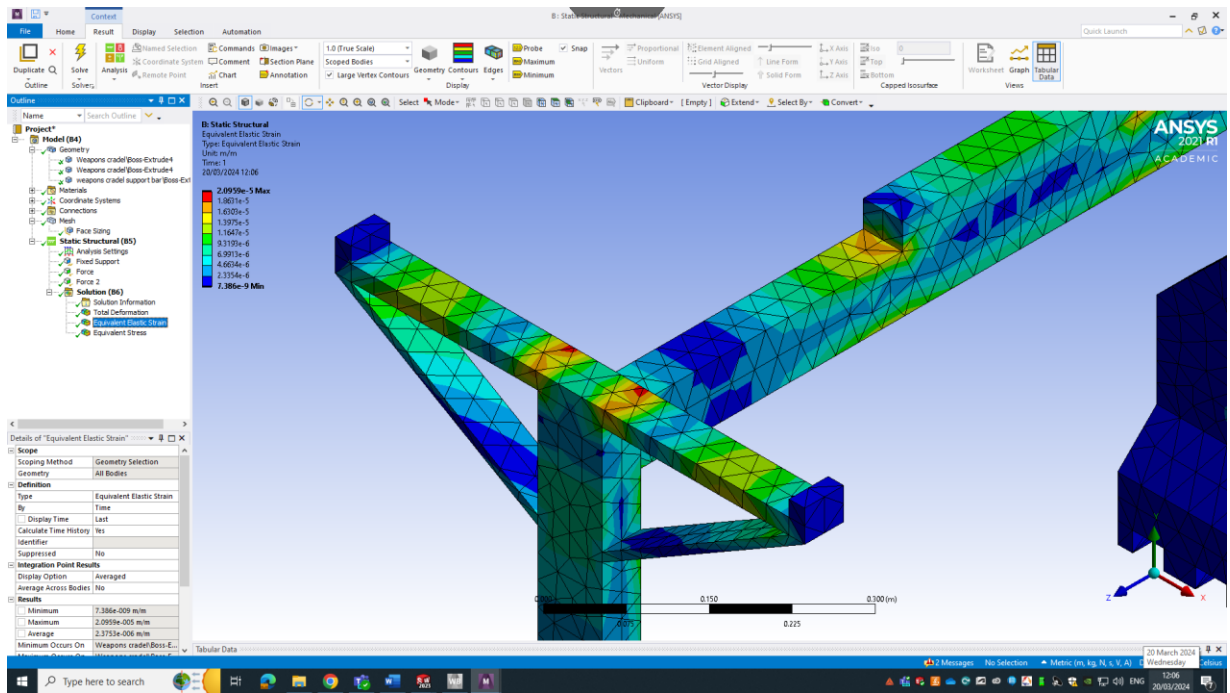


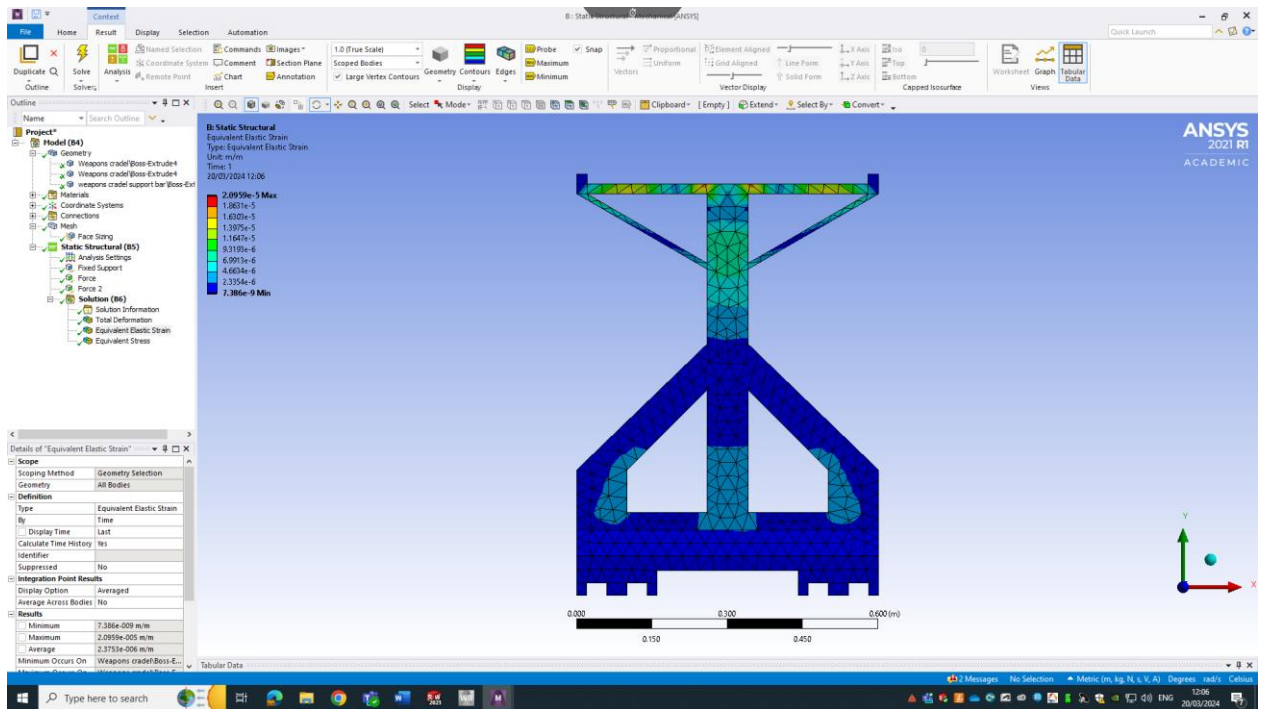




Showing Strain







Showing Stress

