Finite Element Analysis

Coursework 2

GROUP 5

ARCHIE BUXTON, **PDE** (B913210) – **MODEL CREATION AND NX**

BANA QURONFULEH, **PDE** (B920161) – **PRACTICAL TESTING AND ABAQUS**

BENAZ ALADIN, **PDE** (B927700) – **ABAQUS**

EWAN ATKINSON, **PDE** (B821342) – **PRACTICAL TESTING AND VERIFICATION OF RESULTS AND SUPPORT WITH NX**

KHAYA HART, **MECH** (B924463) – **HAND CALCULATIONS AND SUPPORT WITH NX**

YASHVI VADALIA, **PDE** (B915673)– **OPEN SOURCE (FREECAD)**

ALL: INITIAL RESEARCH, PROBLEM DEVELOPMENT, MATERIAL PROPERTIES, EVALUATION OF SOFTWARE, REPORT.

Problem Definition

Aim: The carabiner is rated at a safe working load of 12kN. This project will evaluate this value using the methods mentioned below.

Objectives:

- Model a carabiner using CAD based on an existing product
- Simulate a finite element analysis of the carabiner using Abaqus, NX, and FreeCAD.
- Compare the different software packages and decide which one performed the best.
- Verify results using calculations and in-lab testing.

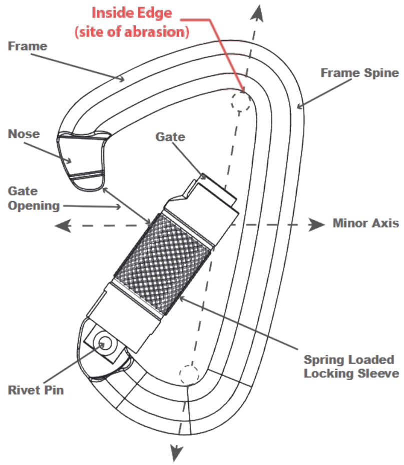


Figure 1: Carabiner Diagram

Problem Context: A carabiner is a standard tool used in climbing and rigging applications. There are many variations of carabiner, including shape, strength ratings, and gate type; each type is designed for a specific application. Despite the extensive literature on carabiner load capacities, several areas still require further investigation.

Carabiner manufacturers test their products to EN 12275:2013 standards for CE and UIAA certification; however, these tests do not always consider how a carabiner may be misused. During a discussion with the founder of RopeLab[1], a company that specialises in testing rigging equipment, they suggested that the primary load-bearing system can occasionally fail during zip-line applications, leaving the safety carabiner to support all the load. As the carabiner travels along the cable, it causes abrasion on the inside edge (Figure 1). This is also a common problem in rock climbing, where rapid rope movement generates friction on the inside edge, leading to wear. However, this problem may go unnoticed if carabiners are left on rock faces for prolonged periods.

As the amount of abrasion increases, it is unclear how much it can withstand before the carabiner must be retired. This project evaluates a carabiner rating to determine how abrasion affects load-bearing using finite element analysis (FEA) simulations. Hand calculations and a practical experiment were conducted to verify these results. It was expected that the carabiner would yield at a lower load as the amount of abrasion increased.

Freskaro's 12kN Screwgate Carabiner Clip[2] (Figure 2) was selected as the test piece for this project. The product specification provided all the information to model the carabiner for the simulations, including the certified rating, dimensions and material. Each test was evaluated to this 12kN rating, which suggests the yield strength of the carabiner.

The simulations were performed using Abaqus, Siemens NX, and non-commercial FEA software (FreeCAD), which feature mesh generation techniques to convert complex geometric problems into discretised elements.

The carabiner is already rated at a safe working load of 12kN. Therefore, this project will evaluate this value using the methods previously mentioned.

1 See details of RopeLab's rigging tests and experiments

2 See Freskaro's carabiner product specification

Assumptions

Dimensions:

Figure 2: Frekaro 12kN Screwgate Carabiner Clip with dimensions

Material Properties:

The carabiners used for practical testing are made of 7075-T6 Aluminium[3]. The following material properties of Al are used for simulations in the three chosen software and the numerical calculations.

- Young's Modulus = 70 GPa
- Poisson's ratio = 0.32
- Coefficient of friction =0.7
- Density = 3 g/cm^3

The yield stress and the corresponding plastic strain value were needed to test when the carabiner would plastically deform in the simulations. A true stress and strain graph (Figure 3) was used to obtain the values (Table 1)

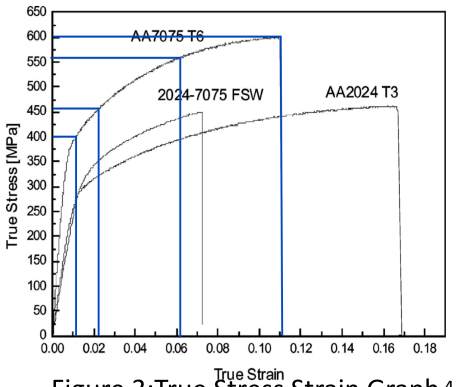
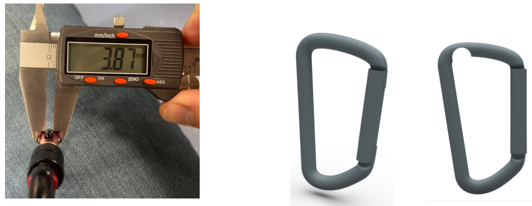


Figure 3:True Stress Strain Graph [4]

Yield Stress (MPa)	Plastic Strain
400	0
480	0.012
560	0.051
600	0.1

Table 1 :Yield Stress



(a) vernier callipers (b) no increment (c) 5mm increment

Figure 4: Modelling

Assumptions made

- Wear concentrated on the inside edge (Figure 1)
- The carabiner is stationary when in use
- Carabiner material – isotropic, Aluminium 7075 as stated in Freskaro's product specification
- No defects in the carabiner
- No surface finish on the carabiner
- Gate is fully locked when the carabiner is in use
- CAD Model: each model features the same geometry but with varying amounts of abrasion on the inside edge (Figure 4b and c)
- Calculations: forces on the carabiner are equal and act vertically in opposite directions, a moment is induced around the Z-axis (which goes into the page) – 2DOF, thickness remains constant at all points, circular cross-section, the carabiner is treated as one solid object (gate and connecting components simplified), the shape is treated as a symmetrical oval, the max internal is assumed to be greater than the max outer stress. The model acts as a curved space beam with clamped supports.
- Testing: Each sample was milled using a 10mm ⌀ drill bit by increments of 1 mm to replicate the effects of abrasion, apart from the control, which featured no abrasion to set the benchmark for comparing the results. 10mm ⌀ loading pins, gate fully locked, 5 mm/min ramp rate, pulled along major-axis

Model Creation:

The same CAD model was used in each software to compare the results accurately. Additional measurements were obtained using vernier calipers, as not all values were provided in the specification (Figure 4a). Additionally, a raster image was used to maintain the shape of the carabiner. The same model was then duplicated five times. Each model featured a 10mm diameter subtraction in the same place by 1mm increments to replicate the effects of abrasion. NX can export the model in appropriate formats for each software (STEP, IGES, and STL). The gate was united with the rest of the carabiner to allow the model to be imported into each software as a single body. In addition, it features an advanced simulation feature, which enables FEA.

---

[3] See Matmatch 7075-T6 Aluminium Properties

[4] See true stress-strain of 7075 Aluminium

Abaqus Discretisation

All six carabiners were simulated in Abaqus using the following specifications, unless stated otherwise. The 'no-hole' model imported into Abaqus can be seen in (Figure 5).

Simplifications to the model

Initially, the carabiner was loaded into Abaqus as two separate parts: the gate and the body of the carabiner. Contacts were added in to connect between the parts as an assembly; however, the contacts failed after the model reached yield. To stop that from happening and to stay consistent with the simulations done on the other software, the carabiner was then loaded in as one fused part.

Additional Modelling Changes

A 10mm analytical rigid pin was added to the assembly at the top of the carabiner and was given a displacement upwards. This created the force needed to deform the carabiner. It was created as an analytical rigid part in order to avoid the pin deforming as we are not assessing the pin. The positioning of the pin was the only discrepancy between simulations with it being placed where was deemed most accurate.

The time step used within Abaqus was set to a time period of 1, an allowance of 1000 incrementations, a minimum increment size of 1E-12, and a maximum increment size of 0.005.

Material used

The material properties used in Abaqus are consistent with the properties stated in the beginning.

Mesh Generation

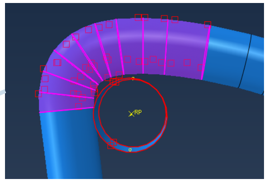
Initially, a global mesh size of 0.5 was used; however, the number of nodes created exceeded the number of nodes allowed by the Abaqus student license. Therefore, a global mesh size of 1 was used since it created a moderate sized mesh. The element shape assigned to the model was Free tet and the type was linear.

BCs

- An ENCASTRE BC was placed at 5 nodes on the bottom of the carabiner to hold it in place.
- A displacement BC was set at the reference point in the analytical rigid pin and was given a displacement on 6mm upwards.

Limitations

- No force can be applied on the pin or the carabiner directly to create the deformation.
- Multiple contacts meant that the simulation was less likely to run, and the results received were not accurate.
- The fixed BC at the bottom showed immediate stress, which was chosen to be neglected.



Contact between pin and carabiner

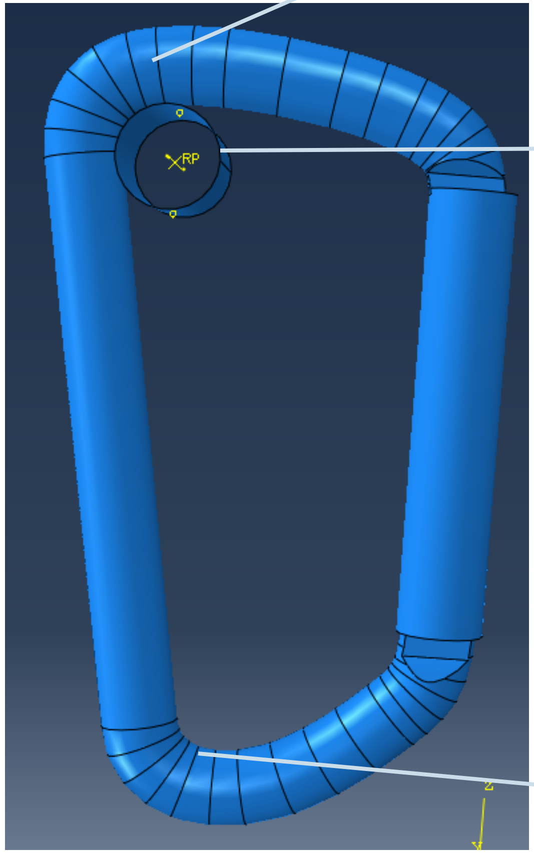
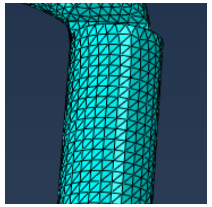
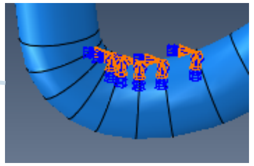


Figure 5: Simulation Set up

Displacement BC upwards



Mesh for entire carabiner – linear free test



Carabiner fixed at the bottom using ENCASTRE boundary condition

Abaqus Results

To find the force acting on the carabiner from the displacement of the pin, the equivalent reaction force (RF) on the reference point of the pin was used. Since the material has an initial plastic yield stress of 400MPa, the RF at that stress was found. This value was then compared to the rated load of the carabiner (12kN).

A second analysis was conducted in order to compare the values obtained for the Von Mises at the top of the carabiner . This is because the other two software used a 12kN load to find the stress, rather than a displacement BC. The RF closest to the value of 12,000kN was found with its corresponding Von Mises stress. The actual value of the RF provided by Abaqus depended on the time step and the different increment inputs, that is why an approximate 12kN was taken.


(s) Bana Quror

Hole Increment Upwards (mm)	0 mm	1 mm	2 mm	3 mm	4 mm	5mm
RF (increment closest to 12000N) (N)	11905.8	11119.80	11174.80	11503.30	11431.40	11901.70
Von Mises at Force (Mpa)	519.04	493.75	547.72	477.33	481.26	491.11
RF at 400MPa Von Mises (N)	4684.78	3767.70	2510.15	3232.40	5124.13	2140.53

Table 2: Abaqus Results

Analysis

Force needed to cause plastic deformation

The force at which the carabiner starts to plastically yield with no holes milled into it was lower than the carabiner rating by 61%. This may be due to the fact the material properties in the simulation set 'initial plastic yield stress (plastic strain=0)' at 400MPa. This value was based on data from a study conducted on the Aluminium material used in this project [4]. However, the physical carabiner is likely to have a higher initial plastic yield stress, allowing it to withstand 12kN loads without plastically yielding.

It was found that with the exception of the 3mm and 4mm increment simulation, the presence of a hole would reduce the force required to plastically deform the carabiner. It was not expected that the force would decrease, then increase again at 3mm and 4mm, then go back down. This could be due to an anomaly in the simulations. However, verification with practical testing (slide 11) showed a similar pattern.

Von Mises at 12kN

The Von Mises stresses were more difficult to assess since the force used to get the stress value was not consistent. The closest RF to 12kN for the 0mm increment was higher than all the other RF values, which could explain why the stress on the top was higher than the rest. However, the pattern at 3 and 4mm was consistent with the results previously discussed. Even though higher stress values were expected, 3 and 4mm showed lower stresses.

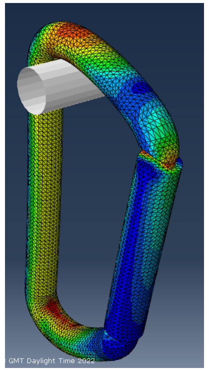


Figure 6: No hole

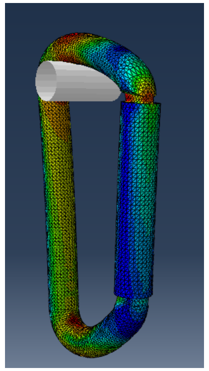


Figure 7: 1mm hole

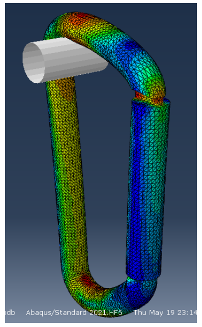


Figure 8: 2mm hole

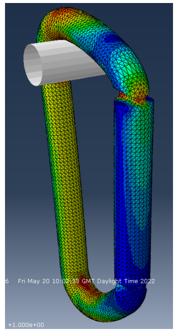


Figure 9: 3mm hole

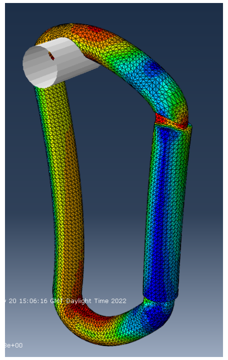


Figure 10: 4mm hole

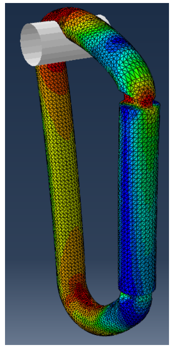


Figure 11: 5mm hole

NX Discretisation

Simulation creation:

- NX Nastran Design in SOL 101 was the solver chosen with a structural analysis type. As the carabiner was only going to be analyzed below its yield strength, a linear statics, single constraint was used.
- Element Iterative Solver was used as it tends to yield faster run times improvements of 4x to 6x compared to the global iterative solver.
- **Material used**:
- 7075-T6 Aluminum was used, (see assumptions for material properties)
- For the Pin (Figure 12) assigned the materials AISI Steel 1008 HR as it was a material used in the jig in the practical verification and therefore deemed to provide a more accurate comparison when compared to the practical numbers.

Mesh Generation:

- CTETRA(10) element meshing type was used as it would provide a second order parabolic tetrahedral element mesh, which would provide greater accuracy on curved surfaces over the linear tetrahedral element mesh of (CTETRA(4). The element size was automatically calculated using a meshing wizard. This gives different meshes for each model ranging between 4.01 – 3.95mm.

Simplifications to the model:

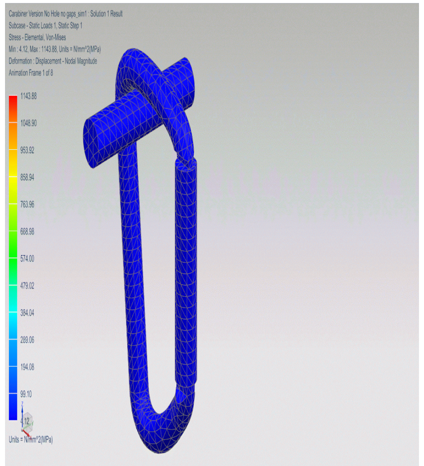
- The gate was united with the rest of the carabiner to create a solid body. This was done to be more in harmony with the FreeCAD (later in the report) and to allow for a better analysis. To provide a more accurate distribution of force over the carabiner a pin was added (Figure 13). This means that instead of the force acting on individual nodes which might have caused inaccuracies as the carabiner deformed, a pin with the dimensions of the rope (10mm) was added, it allows the force to only act on the nodes that act directly above it.

BCs:

- To prevent the pin from moving out the side of the carabiner, a fixed constraint was added to ensure the pin remained in the center of the carabiner the entire time (Figure 14).
- A force of 12kN was applied to the pin in the z axis along the major axis upwards.
- Similar to Abaqus, a fixed displacement was placed on the bottom of the carabiner to prevent the movement of it and to provide the reaction force needed to resist the pins movement.
- Contact faces were made in-between the pin and carabiner to ensure that the surfaces would contact and move each other, the same 0.7 coefficient of friction was used.

Limitations:

- Similar to Abaqus, using a fixed displacement on the bottom of the carabiner caused additional stress and the nodes either side of the boundary condition. This couldn't be fixed as the only alternative was to add a secondary pin, but we were limited by our knowledge on the software and was unable to make it work.



```
Carabiner Version No Hole no gaps_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 4.12, Max : 1143.88, Units = N/mm^2(MPa)
Deformation : Displacement - Nodal Magnitude
Animation Frame 1 of 8
```

Figure 12: NX simulation – no hole

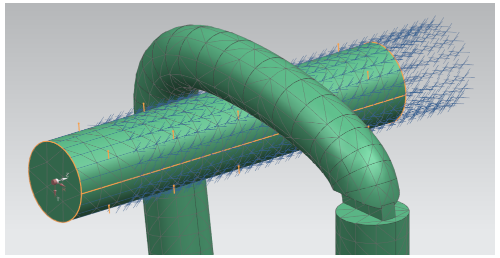


Figure 13: BC top pin

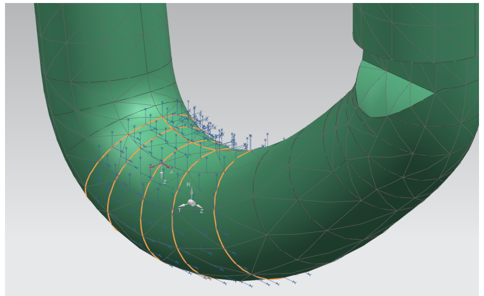


Figure 14: Fixed BC

NX Results

Set up: The displacement and Von Mises were calculated using the liner solution 101, as the calculations would all be done under the imputed material yield strength. Only a slight displacement occurred when the load was applied to the carabiner.

As the size of increment in the carabiner increases, the max Von Mises stress decreases. As the hole increment size increases, the mesh element size decreases. This is because of the need for finer detail. These sizes were automatically calculated, but the range in element size is so small, that they were felt to have a negligible impact on the results.

Each value for displacement and Von Mises were obtained by identifying the values on a singular node to ensure consistency.

Analysis: The values for the Von Mises trend shows a linear decrease apart from the 2mm hole, which has both a significantly higher Von Mises and displacement. It's unknown why this has happened, although further investigation can be done to understand it. This can be seen in Figure 15.

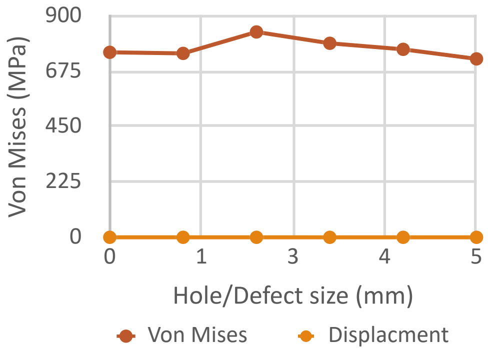


Figure 15 :Von Mises results for each hole size

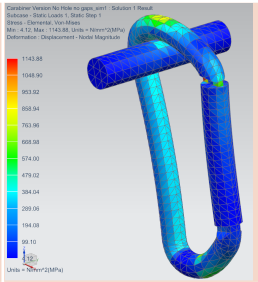
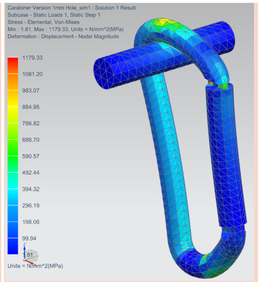
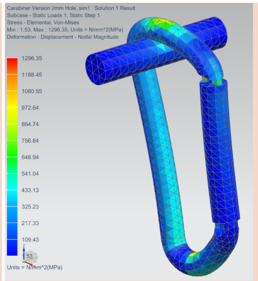
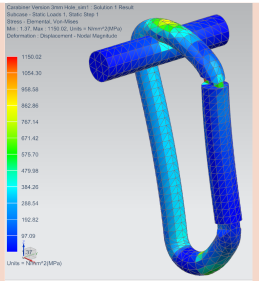
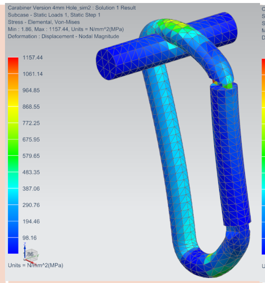
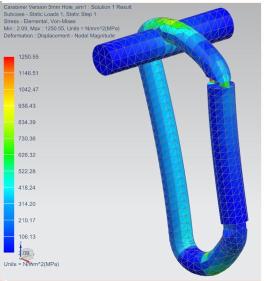
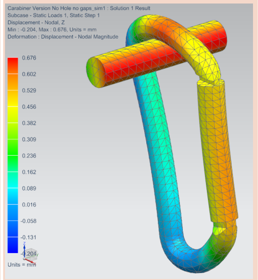
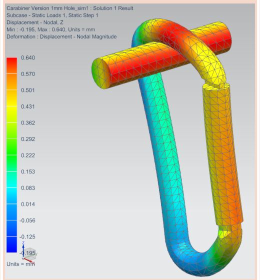
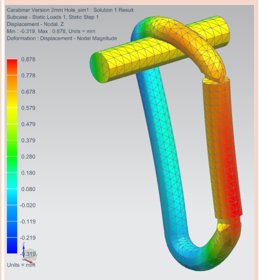
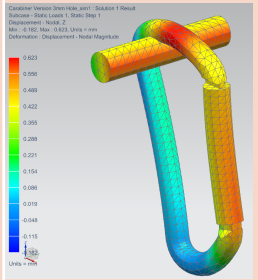
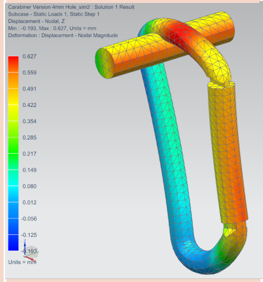
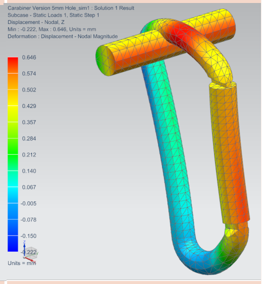
Hole Increment Upwards (mm)	**0 mm**	**1 mm**	**2 mm**	**3 mm**	**4 mm**	**5mm**
						
Max Von Mises (Mpa)	751.846	747.466	834.503	788.506	763.921	725.171
						
Max displacement (mm)	0.531313	0.471892	0.588454	0.533545	0.535425	0.436951
Mesh Element Size (mm)	4.01	3.99	3.98	3.97	3.96	3.95

Table 3: NX results

Open Source (FreeCAD) Discretisation

Two software, Lisa and FreeCAD were used to simulate the carabiner. FreeCAD was chosen to simulate the different defects on the carabiner because it generated better mesh and did not have any limit to the nodes to produce the results.

NX Part file was converted to .stp file which was imported in FreeCAD.

Simplifications to the model:

- The gate and the body were fused as one body (because FreeCAD could not produce results for parts with multiple meshes). This will influence the results as the carabiner and the gate are not different parts and the gate will not open as expected when a force is applied.

Material used:

- 7075-T6 Aluminium

Mesh Generation:

- Tetrahedron Mesh was created using Netgen Mesh from the FEM workbench
- Max element size of 1 was used following what was used in Abaqus and NX.
- The mesh generated was very fine, to produce more accurate results.

For each carabiner with different hole sizes the same BCs were used:

- Fixed support at the bottom faces.
- Force of 12 kN on the top cylinder acting upwards in the z direction.

Limitations:

- Cannot generate mesh of different sizes at different parts of the carabiner. For example, a finer mesh at the high stress points could not be created.
- Carabiner with multiple meshes could not be analysed to give results.
- Plastic strain could not be added to the material properties, because of the which the results produced may differ from those from Abaqus and NX.

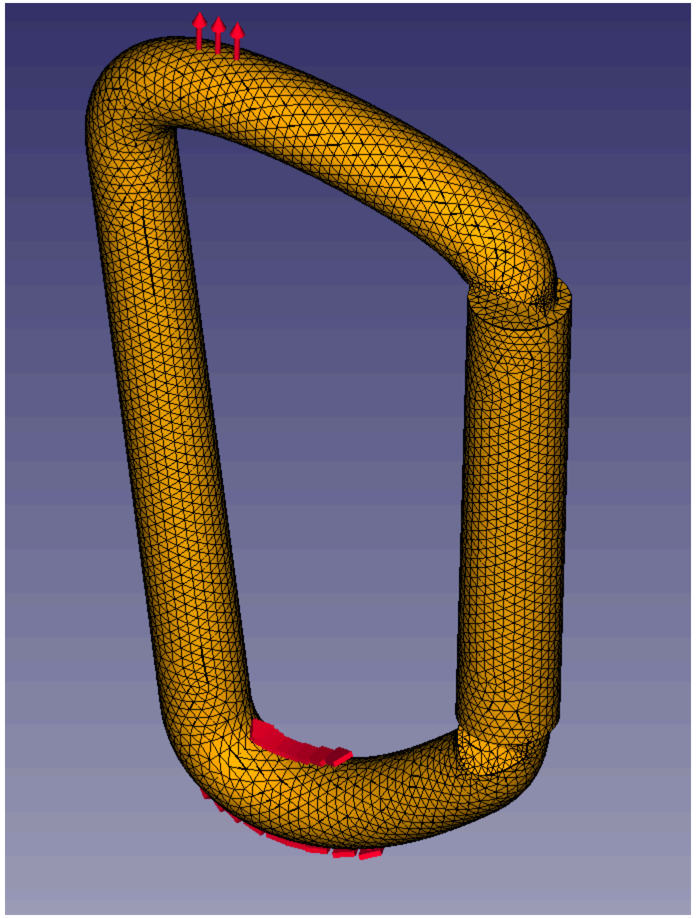


Figure 16: FreeCAD mesh

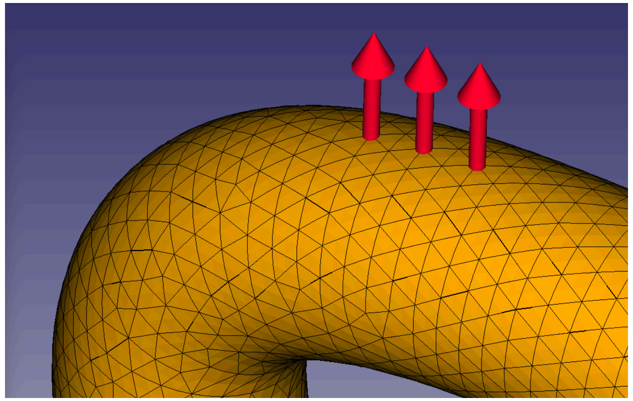


Figure 17: Applied Force

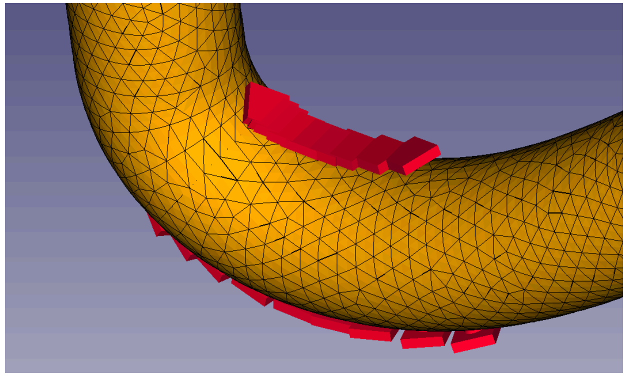


Figure 18: Fixed Support

Open Source (FreeCAD) Results

Hole increment upwards (mm)	0 (Control)	1	2	3	4	5
Von Mises at Top (MPa)	1021.678	848.188	1066.641	1309.91	1695.543	2591.348
Max displacement in z (mm)	0.9	0.87	0.91	1.01	1.12	1.46

Table 4: FreeCAD results

○ None
○ Displacement Magnitude
○ Displacement X
○ Displacement Y
● Displacement Z
○ Peeq
○ Temperature
○ von Mises Stress
○ Max Principal Stress
○ Min Principal Stress
○ Max Shear Stress (Tresca)
○ Mass Flow Rate
○ Network Pressure

Min: -0.25 mm
Max: 0.90 mm

☑ Show
Factor: 0.0
Slider max: 10.0

Preselected: No_hole.ResultMesh. (-45.080299 mm, -3.841970 mm, 75.263298 mm)

Figure 19: Movement of Carabiner with 0 mm hole (control)

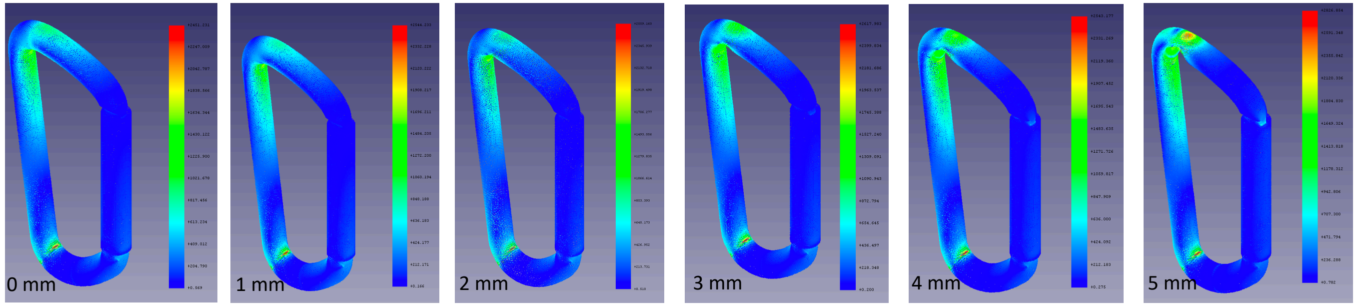


Figure 20: Von Mises Results for each carabiner

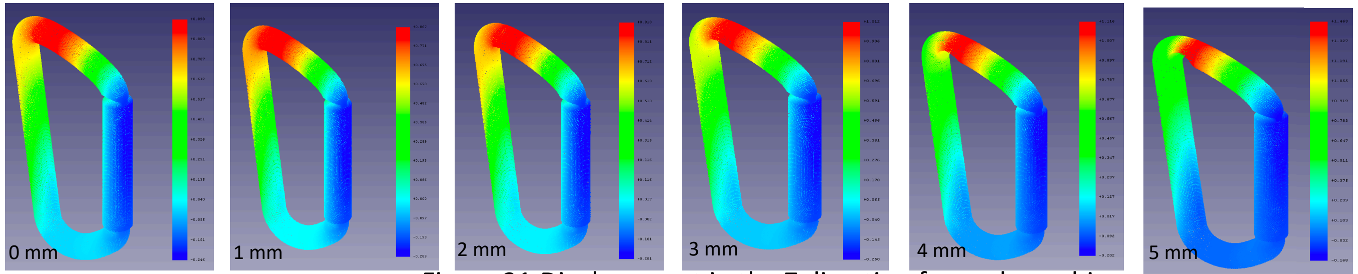


Figure 21:Displacement in the Z direction for each carabiner

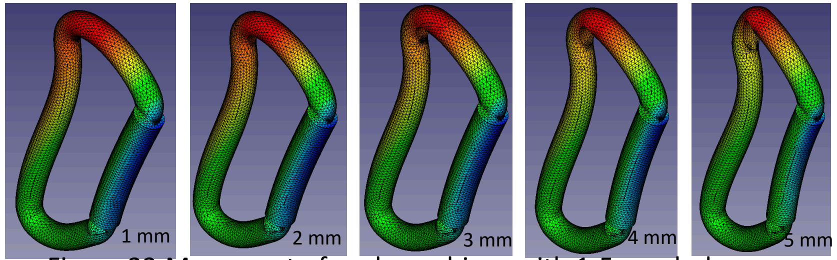


Figure 22:Movement of each carabiner with 1-5 mm hole

Movement:

- All the carabiners showed a similar movement.
- The carabiner with 5 mm hole deflected more than the other carabiners at the same displacement factor.
- Since the deflection is more for the carabiner with larger hole, it is more likely to break quicker than the other carabiners.

Von Mises Results:

- The Von Mises at the top of the carabiner where the force is applied is shown in table 4.
- These increased with increase in the size of holes.
- As expected the highest value was for the 5 mm hole. This region was closer to the maximum Von Mises, unlike the Von Mises values of the other carabiners, which were all around the middle region of the spectrum.
- Even the Von Mises values at the nose increased for the 3 mm, 4 mm and 5 mm holes.

Displacement (z direction) Results:

- For all the carabiners, the highest displacement was where the force was applied.
- The displacement values increased with increase in the size of holes.
- However for the 1 mm hole, the displacement value reduced from that produced for the control.

Software Comparison against Abaqus

Abaqus was considered the most professional software of the three, and therefore a comparison against Abaqus was created:

Advantages	**Disadvantages**
Abaqus has a much larger variety of capabilities and was the only software in which the pin was successfully used in a displacement method which was how the real-life validation was carried out.	Both NX and Free CAD have a much simpler interface to use; this led to much faster learning of how to use the software. Abaqus's interface is outdated and overcomplicated. This meant that despite the group having the tutorials, an understanding of how to accomplish what was needed was difficult to obtain.
It was easy to import the model into Abaqus without any alterations and errors.	With all the added complications in Abaqus, it meant that solving the job took significantly more time to complete when compared to NX and Free CAD. This might have been for each software's iterative solvers, allowing them to solve faster. For example, NX's Element Iterative Solver was used as it tends to yield faster run times improvements of 4x to 6x.
Meshing the model was simple even with its complex geometry.	For Abaqus, the NX model had to be formatted into a STEP file before it could be imported. This means no other parameters could be added, such as materials, as the format focuses on the form of the model. It is also challenging to change the model once imported and thus very slow to adapt to changes. This is also the same for Free CAD but not for NX, as the model can be changed and altered with updates propagating through the FEA application of NX.
Even though it was not possible to add a load onto the carabiner, it was the only software that showed the stresses at different forces.	

Table 4: Advantages and Disadvantages of Abaqus in comparison with NX and FreeCAD

Overall, despite all the complications of learning how to use Abaqus, it was decided that it was the better software for this project.

Practical Testing

Introduction and Methodology:

- Practical testing was conducted to verify the results of the simulations.
- After milling each sample (Figure 23), the carabiners were placed in a vice that locked them along the major-axis (Figure 1) for consistency. A jig was used to prevent slippage during the milling process
- The samples were loaded into a tensile test machine that operated at a ramp rate of 5mm/min; an extensometer measured elongation in the material
- Hypothesis: the carabiner's yield strength will reduce as the abrasion increases

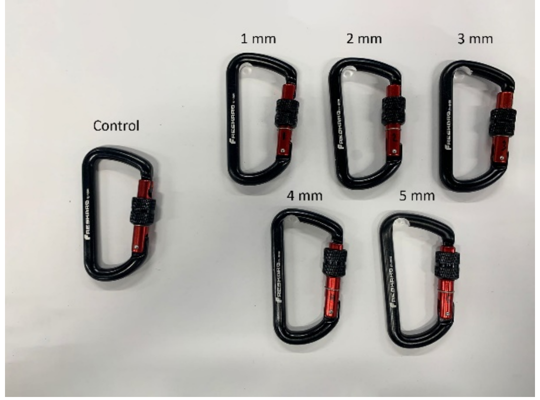


Figure 23: Carabiner testing

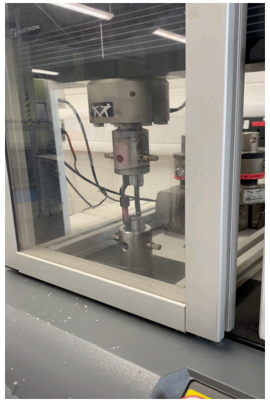


Figure 24: Carabiner tensile testing

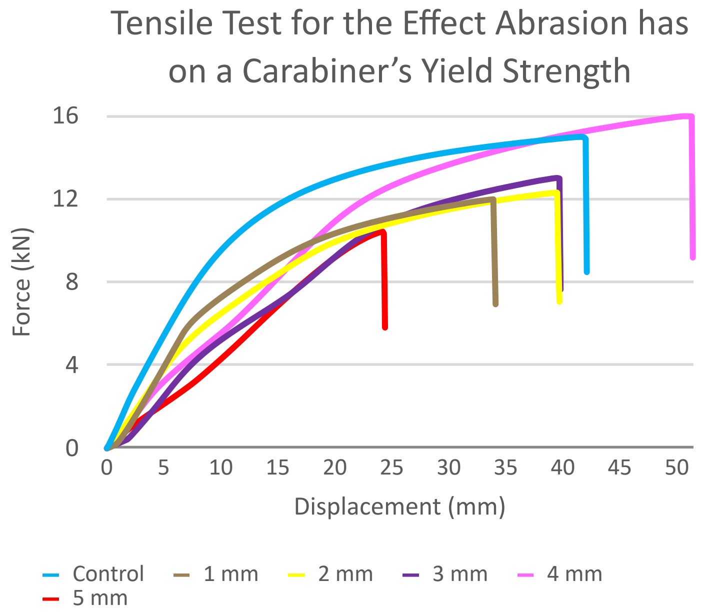


Figure 25: Tensile Test for the effect abrasion has on a carabiner's yield strength

Results:

- Every experiment surpassed the stated yield strength (12kN), except for 1 mm and 5 mm, which reached their ultimate tensile strength (UTS) at a lower value
- The data in Figure 25 shows no clear trend; therefore, the hypothesis cannot be proven. The data suggest an initial drop in yield strength as soon as abrasion occurs; however, the yield strength is restored after 2 mm (Figure 29).
- Table 5 demonstrates observations from the practical test. The 5 mm experiment had the lowest yield strength and failed at the site of abrasion, as expected; however, the others failed at the gate (Figure 24) or along the major-axis
- Additionally, the average UTS (13.07kN) was higher than the stated value (12kN). From a conversation with RopeLab, they suggested it is common for manufacturers to state a lower value than what it is capable of; therefore, the minimum value is taken rather than average
- These results imply other variables affect the yield strength. Further experiments are required to assess how abrasion affects its performance more accurately. These experiments should evaluate different types of carabiners, including higher-rated carabiners and alternate designs (shape, cross-section, gate type)
- Limitations:
  - The experiment used metal loading pins rather than rope; therefore, it is not true to the practical application
  - The 12kN carabiner is not intended for high load applications

Sample	Condition at 12kN	Failure Location
Control	Plastically Deformed	Gate
1 mm	Reached UTS	Gate
2 mm	Plastically Deformed	Major-axis
3 mm	Plastically Deformed	Gate
4 mm	Plastically Deformed	Major-axis
5 mm	Reached UTS	Inside edge

Table 5: Tensile testing observations

Numerical results

Initial Considerations

- Possibility to simplify into a multistage problem.
- Could simplify into several straight-line segments.
- Simplification of local areas was discussed in **slide 3** and included assumptions such as: treating the body of the carabiner as 1 solid shape, circular cross section and changing the shape to be symmetrical and oval. Appropriately simplifying the problem allowed sufficiently accurate and quality results to be obtained, as well as aligning better with the FEA findings.

Mathematical Approach

- Euler–Bernoulli's beam theory seemed the most suitable calculation approach for the application of the carabiner. Beam theory investigates the load-carrying and deflection characteristics of beams. Online sources also appear to use this method when dealing with similar structures. Due to the carabiner's shape curved beam theory was used, which considers features such as the radius of curvature of the carabiner. This method calculates the max stress from the addition of axial and bending stresses present in the carabiner, at its force rating (max force). The exact equations used are shown to the right of this slide.
- A separate approach, using the theory of elasticity was investigated. This theory treats the relationship between forces applied to an object and the resulting deformations. The values obtained from this method agrees well with the curved beam theory. Sources found did not appear as suitable as the beam theory sources and so this method was only used for theoretical purposes. The max stress obtained from this method was 3.1GPa.
- For the damaged carabiners fracture/crack mechanics were used. The abrasion damage was simplified to its most appropriate model, which was a 'thumbnail crack in a solid cylinder'. This allowed the new max stress, due to the abrasion, to be calculated on the carabiner at the force rating. Some of the factors this method considered included: the depth of the abrasion/crack, the surface area of the cylinder and the axial and torsional stresses previously calculated. The results obtained imply that the damage directly affects the max stress at the 'neck'. The FEA software shows lower results, likely due to the location of the wear, which represents abrasion more likely seen in the real world, I.e. due to friction from the rope. The exact equations used are shown to the right of this slide.
- The global variables and properties used are listed to the right of the slide. Figure 26 shows where these values relate to the carabiner.
- Table 6 shows the max stresses at each crack size.

Global Dimensions and Properties

- Force rating = F_{max} = 12kN = 12000N
- Radius of curvature (center of beam) = R_c = 17.5mm
- Diameter = D = 8mm
- Radius = D/2 = 4mm
- Surface area = A = π x R^2 = π x 4^2 = 16πmm^2

Curved Beam Theory

- Stress axial = σ_a = F/A = 12000/16π = 239MPa = 0.2GPa
- Stress bending (I - 2nd moment of area) = σ_b = (F x R_c x R x 64)/(π x D^4) = (12000 x 17.5 x 4 x 64)/(π x 8^4) = 4178MPa = 4.2GPa
- Stress max = σ_{max} = $\sigma_a + \sigma_b$ = 239 + 4178 = 4417MPa = 4.4GPa

Thumbnail Crack Analysis

- Size of crack = c = 1 to 5mm (1mm increments and c = 1mm for example hand calculations)
- Constant B = (π/2) x (c/D) = (π/2) x (1/8) = π/16
- Constant H = 1 – sin(B) = 0.805
- Constant G = 0.92 x (2/π) x (1/cos(B)) x $(\tan(B)/B)^{1/2}$ = 0.601
- Geometry factor, tension = Y_t = G x (0.752 + 1.286 x B + 0.37 x H^3) = 0.720
- Geometry factor, bending = Y_b = G x (0.923 + 0.199 x H^4) = 0.605
- Stress intensity factor = K = (Y_t x σ_t + Y_b x σ_b) x $(\pi \times c)^{1/2}$ = 4785MPa = 4.8GPa

Crack size (mm)	Max stress (Gpa)
0	4.4
1	4.8
2	7
3	10
4	14.2
5	22.4

Table 6: Calculation results

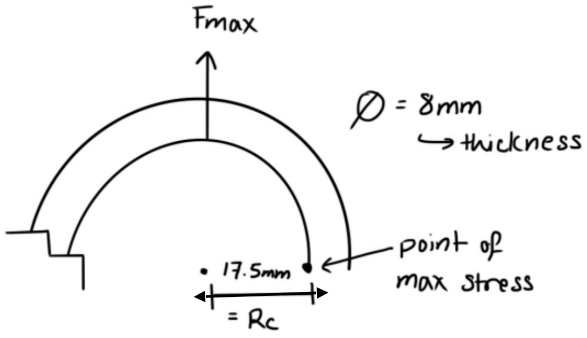


Figure 26: Representation of global properties and dimensions

Results Verification

Commercial to Open-Source FEA Simulation

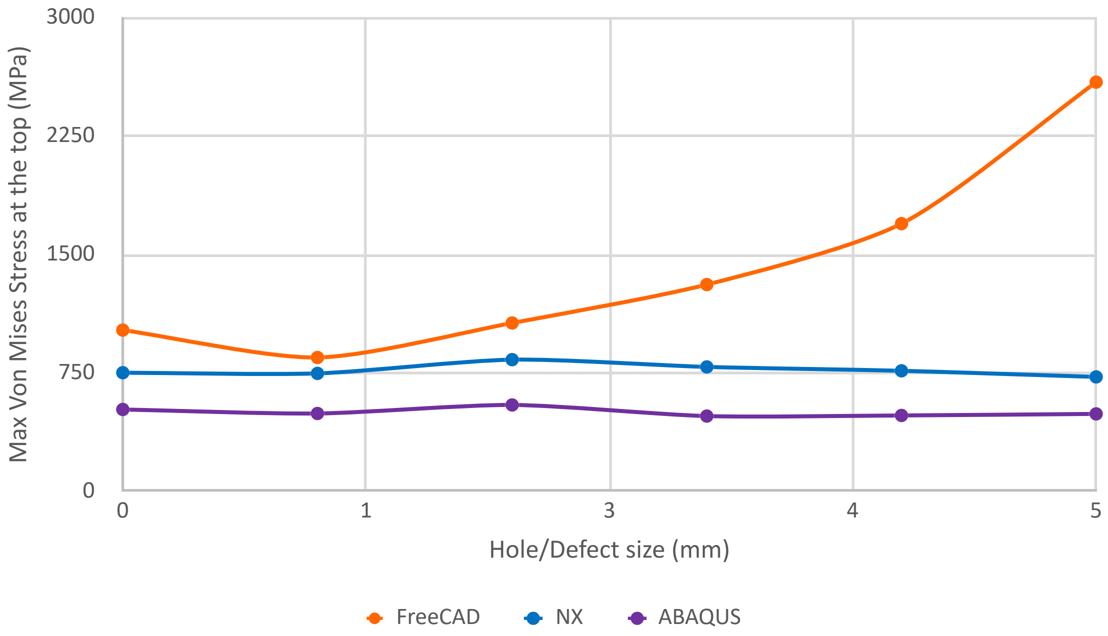


Figure 27: Comparison of Von Mises results between software

- The Von Mises values generated by all the software for each of the carabiner were compared.
- While values generated from NX and Abaqus followed a similar trend, the values from FreeCAD showed an increasing trend from 1 mm increment onwards. The difference is likely because the plastic strain was not considered for FreeCAD.
- The difference of values produced from NX and Abaqus could be because the force was applied directly on the pins for NX, and displacement was applied on the pins for Abaqus.

FEA Simulation to Hand Calculations

Hole Size (mm)	FreeCAD (MPa)	NX (MPa)	Abaqus (MPa)	Calculations (MPa)
0	2394.17	1357	676.082	4400
1	2459.69	1328.3	629.564	4800
2	2509.45	1232.78	579.544	7000
3	2389.89	1295	651.905	10000
4	2402.36	1304.35	565.934	14200
5	2544.32	1368.56	390.663	22400

Table 7: Comparison of maximum principal stress values

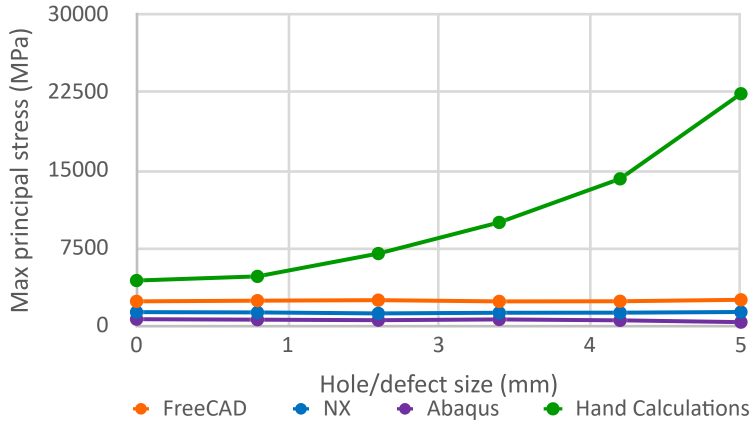


Figure 28: Comparison of Max principal stress between software and hand calculation

- The maximum principal stress values obtained from all the three software were compared with the hand calculated values.
- The hand calculated values showed an increasing trend, while the ones produced from the software didn't follow any trend as such.
- The values calculated were much higher than the values generated from the software. This may be because of the way the force is applied. On the software the force is applied on the defect and for hand calculations it is not.

Results Verification

FEA Simulation to Practical Testing

Abaqus provided the most representative set-up compared to the practical simulation, making it the chosen software to compare the physical and virtual data. Furthermore, the data from Abaqus could be exported the same as the practical test; therefore, stress-strain and yield point graphs were developed to compare them.

While the data sets in Figure 29 do not overlap, they follow the same overall trend. The yield strength in each test initially reduces as the abrasion increases; however, it recovers from 3 mm to 4 mm, then drops off again at 5 mm. This result may be due to the load acting over a larger cross-sectional area (Figure 30); however, further testing is required to validate this prediction.

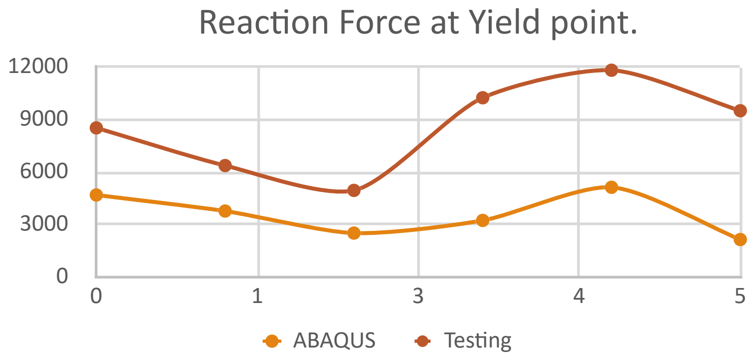
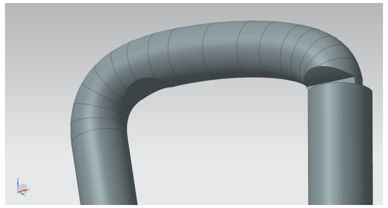


Figure 29: Reaction force at yield point comparison between Abaqus simulation and practical testing

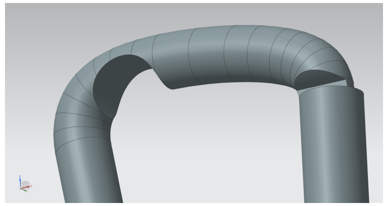
Figure 31 compares the apparent stress and strain curves from the Abaqus simulation and practical test. These graphs allow a more accurate prediction of the behaviour of geometry. The stress results for the simulation were higher overall; however, this is likely due to Abaqus not simulating the UTS.

The results demonstrate a positive correlation between the experiment and simulation in the elastic and the plastic regions; however, the carabiner's yield strength was higher during the practical test, as demonstrated in Figure 29.

It should also be pointed out that the shape of deformation of each model closely resembles the practice testing, thus further confirming the accuracy of modelling.

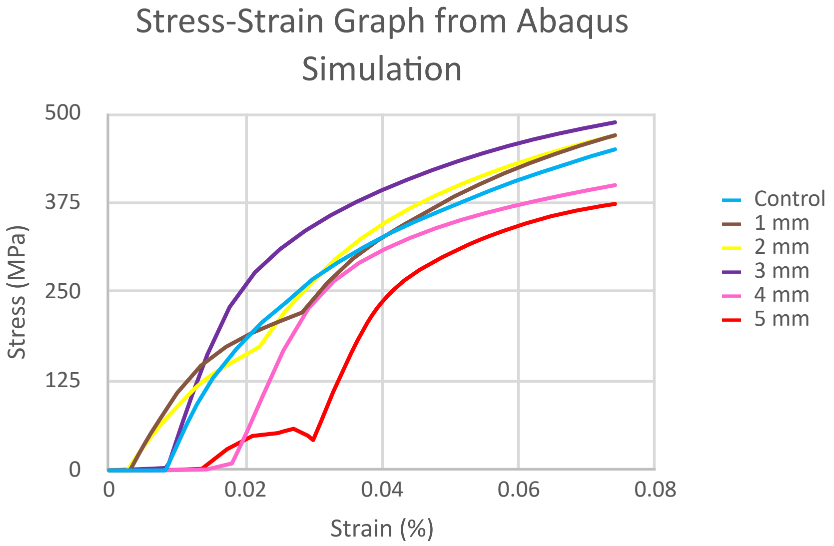


a) 1 mm abrasion

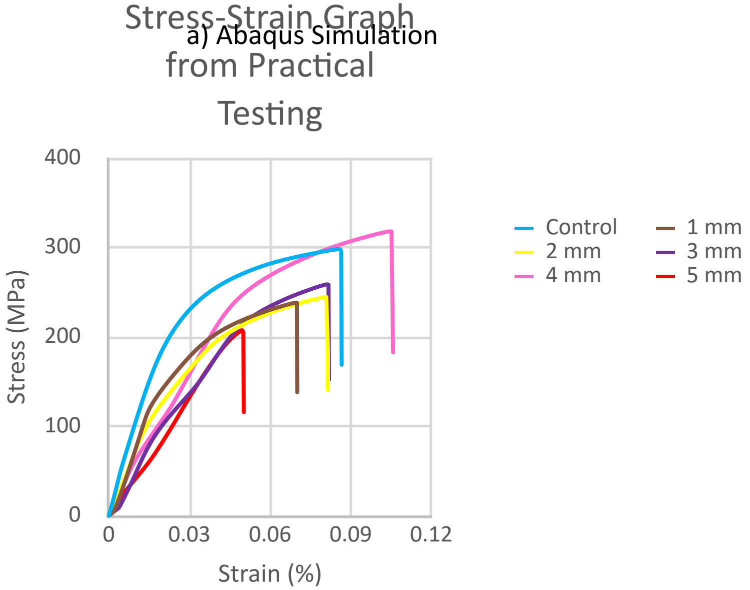


b) 5 mm abrasion

Figure 30: Comparison of cross-sectional area at sit of abrasion



a) Abaqus Simulation



b) Practical testing

Figure 31: Stress-strain comparison between Abaqus simulation and practical testing

Conclusion and Improvements

Conclusions

Practical Testing

A practical test was conducted to investigate the real-world yield strength of each carabiner. The experiment results were inconsistent, as the carabiner failed at different locations regardless of the amount of abrasion, and the yield strength showed no correlation to the hypothesis. Despite this, a comparison between the practical test and Abaqus simulation suggested a similar trend in the yield strength depending on the amount of abrasion; therefore, further testing is required to understand additional variables which are likely the cause of the inconsistencies.

FEA Simulation

This project used a stated carabiner rating (12kN) to compare the results between FEA simulations, practical testing and hand calculations. Abaqus, NX, and open-source software were used to simulate the experiment. A range of open-source software was researched before considering Lisa and FreeCAD. Both these software packages were trialled before choosing FreeCAD.
On Abaqus, the simulation requires a pin that is given a displacement BC and a fixed constraint applied directly to the carabiner. The force at which the carabiner initially starts to yield plastically was obtained. NX also has a pin but had an applied force on it instead. The fixed constraint was also applied directly to the carabiner. The Max Von Mises and displacement were obtained for the rated 12kN force.
FreeCAD had a force and constraint applied directly to the carabiner. Then the Max Von Mises and displacement were also obtained for the rated 12kN force.

Hand Calculation

The hand calculations required several assumptions and means of simplification. These aimed to largely match the CAD models, which allowed for comparison of these results with the FEA and practical testing. The max stress achieved in the hand calculations was greater than the software, most likely due to the simplifications required. For the hand calculations, the joint/pin connection point could be analysed. The direct stiffness method would be suitable for this and the effect the force has on the mechanism could be investigated.

Overall, this project has evaluated the safe working load of a carabiner. As well as successfully comparing understanding some of the limitations of FEA software's.

Improvements

Practical Testing

The carabiners were milled to replicate the effects of abrasion; however, this was a simplification as recreating it would be impractical. Further analysis may consider different sizes and shapes for the abrasion site, depending on how it affects a carabiner.

Furthermore, the test was not truly representative of the practical application identified in the problem definition. Metal loading pins were used rather than a rope; therefore, the stiffness of the pins would have likely skewed the results. Additionally, the carabiner chosen for these tests is not intended for high-load applications. Future studies should consider using a carabiner that would be used in a zip-line or rock-climbing application.

FEA Simulations

A uniform mesh was used throughout the carabiner for all the software. A non-uniform mesh could be created, with a finer mesh at the contact points and higher points of stress.
The gate was united for all the simulations, which does not align with the use of carabiners in real-life. To improve the results and get them as close to the real-life application, more advanced software could have been tested. A software that allowed a simulation with contact between the gate and frame to run (Figure 32), rather than modelling it as one part (Figure 33) may have led to more accurate results.

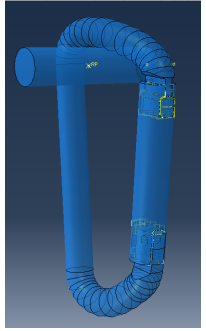


Figure 32: Use of contacts between gate and frame

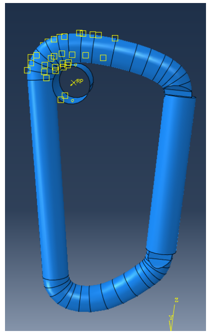


Figure 33: No contact – one part