Comparing Real world measurements with Finite Element Results

Assessment 02

20-MAY-2020

Tony Pauly

ID – 19490954

Table of contents

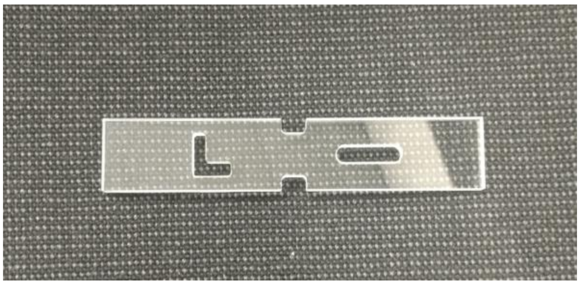
List of figures

List of tables

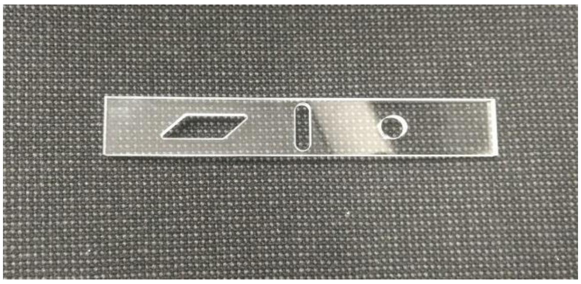
1. **Aim**: - The aim of this assessment is to compare the finite element analysis using solid works and the experimentally tested samples value in the laboratory.
2. **Introduction**: - Solid work is a software for designing the mechanical parts and runs the designed samples and comparing the results with the real-world conditions. So, here in this experiment there are three different acrylic samples of dimensions 150mm*25mm*6mm is cut using laser cutting method and tested experimentally by applying an external load through clamping. These clamps are placed on given area of one side of the specimen. After clamping, an external load was applied on the given area of the sample until it breaks. The values and position of sample failure are noted. After that, these samples are drawn in the solid work software with the same dimensions and runs under finite element analysis at a given boundary conditions. After that, the samples were compared.
3. **Method:**

1.1 Laboratory Method

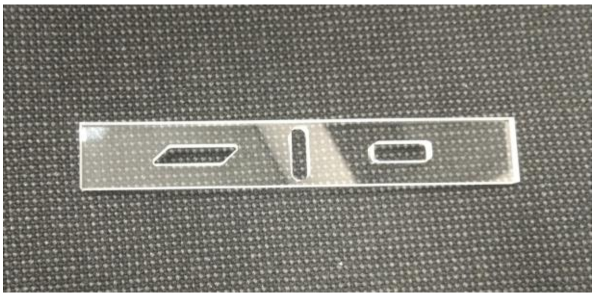
Here in this experiment, acrylic material is cuts in to three different set of samples using laser cutting method at a given dimensions of 150mm*08mm*10mm which shown in the figure 1. Then these samples are tested by applying external load on desired point, until it breaks. The applied load and breaking points are noted and compared with the finite element analysis using the solid works. However, because of the various shapes cut out from the slabs each piece is different from the other.



Sample 05



Sample 08



Sample 10

Figure 1: Three samples acrylic material

3.1.1 Sample 05

As shown in the figure 2, the sample was attached at the end, depth of 20mm along the 6mm x 150 mm face. After fixing the specimen, an external load is applied to this specimen of 30mm from the free edge as shown in the figure 2 and the load is applied till the sample breaks.

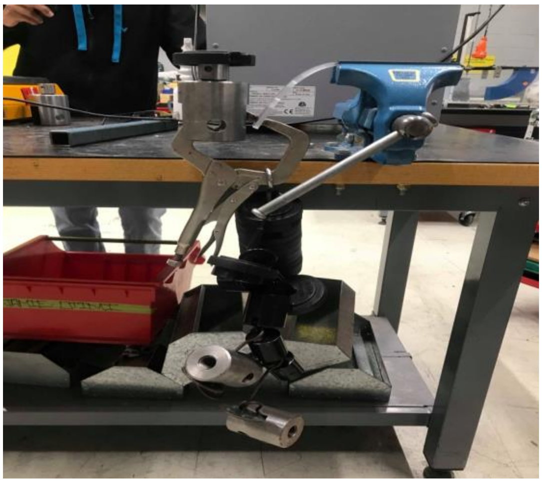


Figure 2: sample 05 laboratory method

3.1.2 Sample 08

In the case of sample 08 the specimen is clamped at the end, depth of 20mm along the 6mm x 150mm face as shown in the figure 4. Then the external load is applied to the sample of 30 mm form the free end of the 150mm x 10mm face.

Figure 3: sample 08 laboratory method

3.1.3 sample 10

In the sample 10 is loaded at the middle of 75mm from the either ends of the flat surface of the specimen. Also, the spacemen are fixed like simply supported at a distance of 10mm from the flat surface, which is opposite to the load applied to the flat surface.

Figure 4: sample 10 laboratory method

Experimental results are shown in the table below

Table 1: Three sample Results

Sample name	Force(N)	Force location	Fixture type
Sample 05	183.38	30mm from the free end	Cantilever
Sample 08	27.05	30 mm from the free end	Cantilever

Sample 10	82.22	75 mm from either end	Simply supported beam

1.2Finite element analysis Method

Firstly, download the acrylic sample from the Moodle. Them create a finite element analysis model by using the solid works. Also, in the Finite element analysis should assign the same load and boundary conditions from the laboratory method and assigning the values. Then the tree different type mesh should apply and note down or record the output result. This method is explained below step by step.

1.2.1 Sample 05

Download the file from the Moodle. Browse the file in the downloads and open it by using solid work software. A sample 05 model will open which is shown below. Here, in the finite element analysis, there are three different steps are required to get the results and these are explained below.

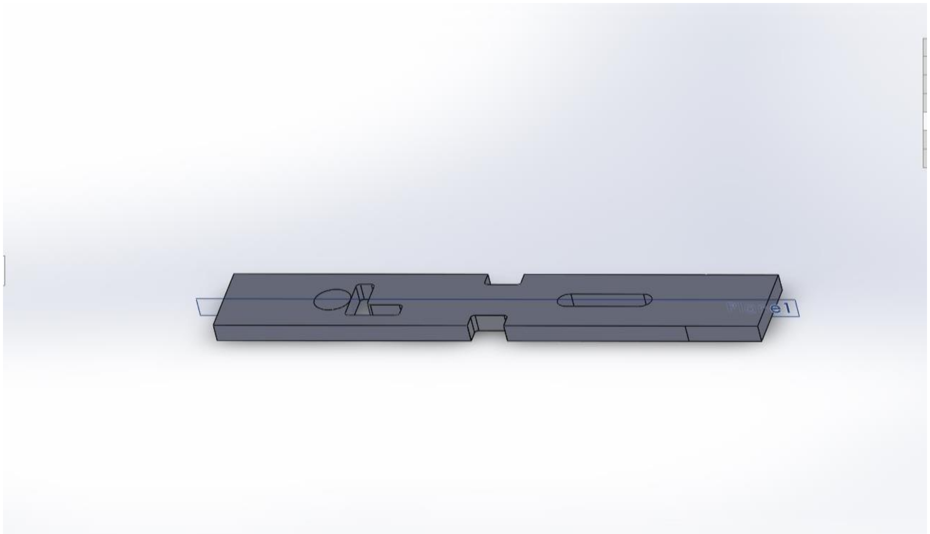


Figure 5: sample05 model

Step 1: Applying the acrylic material into the sample05

- After opening the file in the solid work, click on the simulation tab which is shown in the top side of the software, then a new set of icons will appear. Then click the study, new study. Assign the name it as a static 1 and click ok.

- After this, the next major step is to select the material. In the feature manager design tree, right click on to the sample05 and click on apply/edit material. Then, select the material as acrylic, click apply.

Step 2: Assigning the same boundary conditions to the sample05

- Apply the fixture to the material of 20mm from the free end of 6mm x 150mm face, which is shown in the figure 8.
- Apply load of 183.38(N) to the material of 30mm from the free end of the flat surface, by right clicking the external load in the feature manager design tree. Which is shown in figure 9.

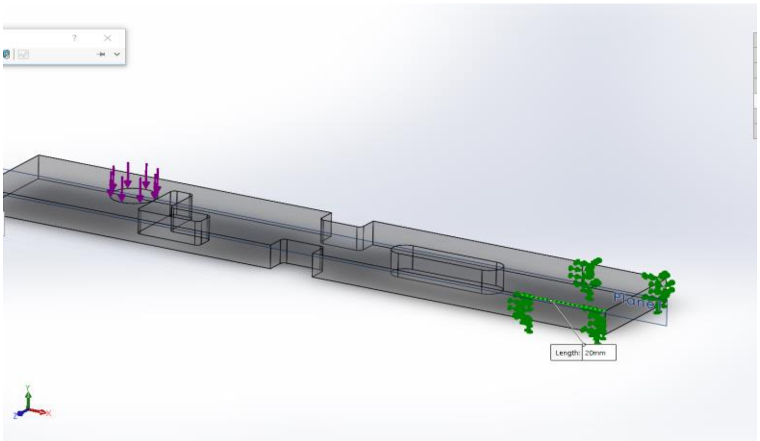


Figure 6: applying fixture on sample 05

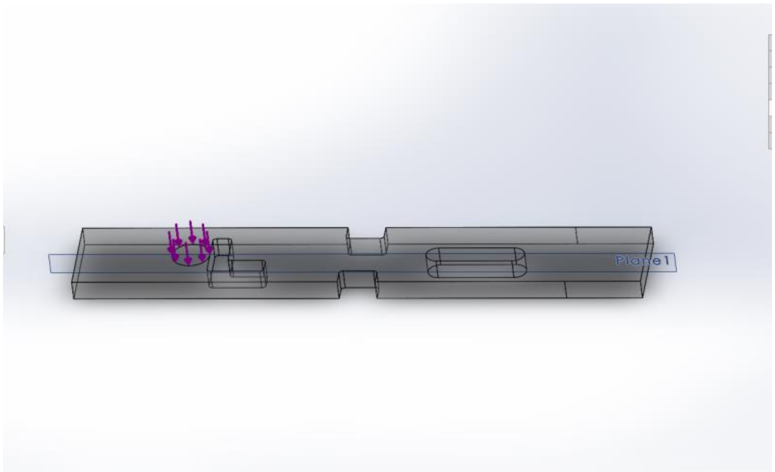


Figure 7: applying load on sample 05

Step 3: Applying the three different mesh

- Here we are giving three different type of mesh by right clicking the mesh on the features design tree. Also, these three different Mesh should run in three different studies with same boundary conditions by right clicking the static 1 and click the copy study and then assign the reimaging two mesh details.

Mesh Details

Study name	Static 1 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	3.27311 mm
Tolerance	0.163655 mm
Mesh quality	High
Total nodes	8937
Total elements	5032
Maximum Aspect Ratio	9.2457
Percentage of elements with Aspect Ratio < 3	98.3
Percentage of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:01
Computer name	

Figure 8: Mesh 1

Mesh Details

Study name	Mesh 2 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	2.11388 mm
Tolerance	0.105694 mm
Mesh quality	High
Total nodes	25770
Total elements	15813
Maximum Aspect Ratio	7.884
Percentage of elements with Aspect Ratio < 3	99.6
Percentage of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:02
Computer name	

Figure 09: Mesh 2

Mesh Details

Study name	Mesh 3 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	0.8 mm
Tolerance	0.04 mm
Mesh quality	High
Total nodes	386338
Total elements	266662
Maximum Aspect Ratio	17.312
Percentage of elements with Aspect Ratio < 3	99.8
Percentage of elements with Aspect Ratio > 10	0.00188
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:10
Computer name	

Figure 10: Mesh 3

3.2.2 sample 08

This is the image of the sample 08, which is download from the Moodle and opened in the solid work software, which is shown in the figure 13.

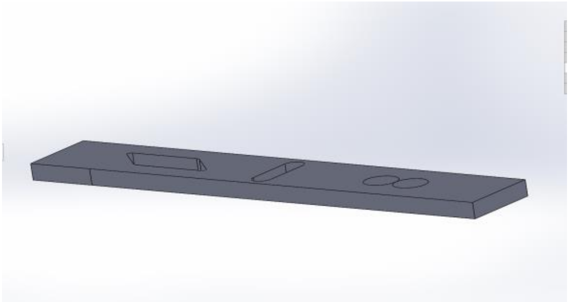


Figure 11: sample 08

Step 1: Applying the acrylic material into the sample08

- click in the simulation tab, then click on new study. Then click ok.
- Right click in the sample 08 in the feature design tree and click on apply/edit material. Then select the acrylic material by clicking the plastic section. Then click on apply.

Step 2: Assigning the same boundary conditions to the sample08

- Next step is to apply the fixture of 20mm from the free end of 6mm x 150 mm face by right clicking the shown in the figure 14.
- Apply loads of 27.05, 9, 18 Newton of 30mm from the free end of the flat surface in a three separate studies, however the fixture are remains same.

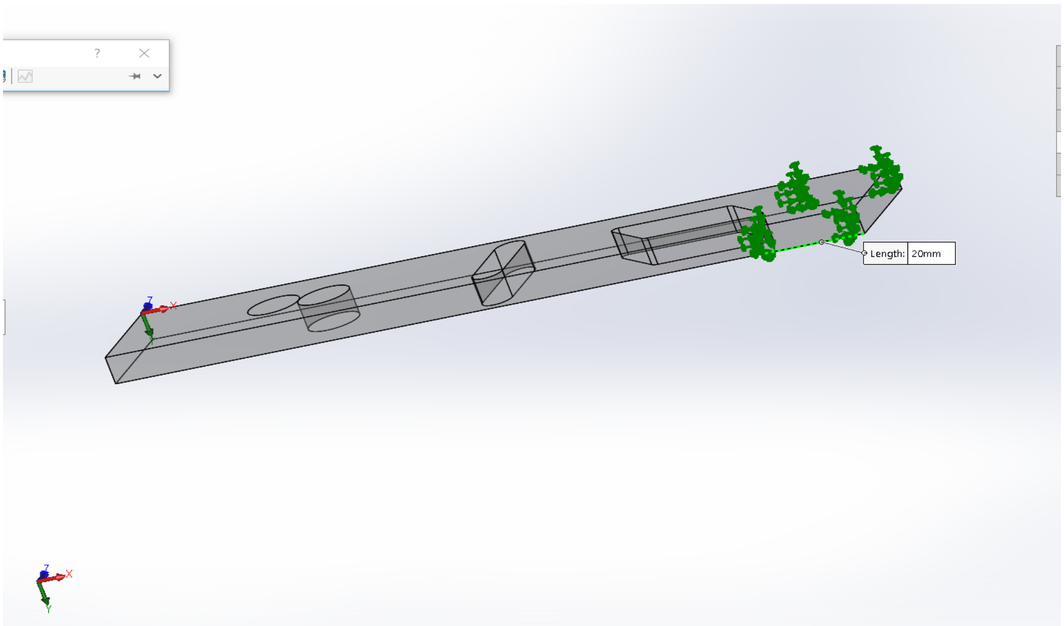


Figure 12: applying fixture on a free end

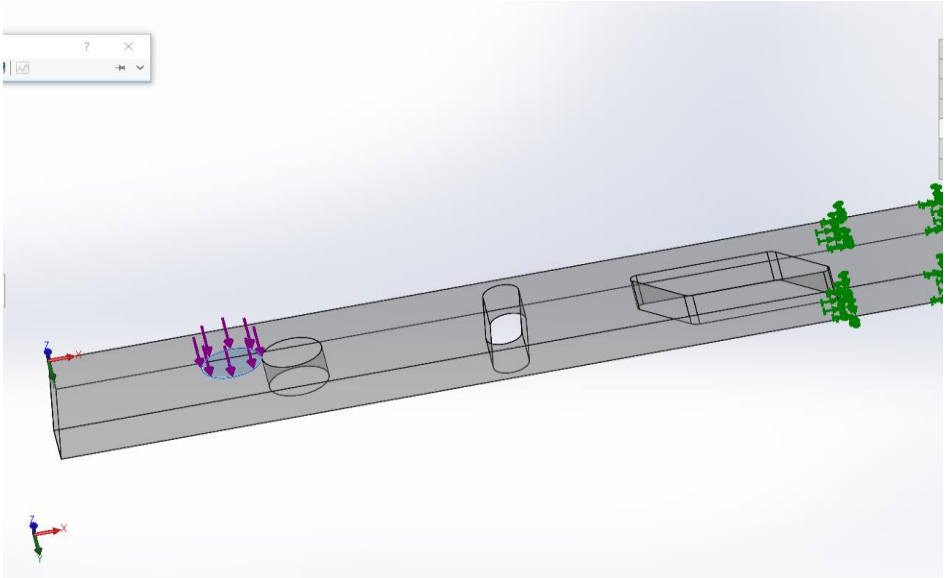


Figure 13: applying load of 27.05,09,18(N) in three different studies

Step 3: Applying the three different mesh

- Here in this sample, there are three type of mesh. So, these three different types of mesh are applied to the sample 08 load 27.05. Hence, we need to copy the study of static 1 by right clicking the static 1 and click copy study. Name is as the mesh 2 and 3. Here in the mesh 3 is taken as the fine mesh of 0.5 as shown in figure 14,15,16.
- As in the case of the load 18 and 9 N, take it as the fine mesh of 0.5 as shown in the figure 17 and 18.
- Then click run study.

Mesh Details

Study name	Static 1 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	2.83895 mm
Tolerance	0.141948 mm
Mesh quality	High
Total nodes	11781
Total elements	6815
Maximum Aspect Ratio	11.124
Percentage of elements with Aspect Ratio < 3	99.2
Percentage of elements with Aspect Ratio > 10	0.0293
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:01
Computer name	

Figure 14: static 1 (course mesh)

Mesh Details

Study name	Mesh 2 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	1.89263 mm
Tolerance	0.0946317 mm
Mesh quality	High
Total nodes	31408
Total elements	19338
Maximum Aspect Ratio	6.7695
Percentage of elements with Aspect Ratio < 3	99.7
Percentage of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:02
Computer name	

Figure 15: Mesh 2

Mesh Details

Study name	Mesh 3 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	0.5 mm
Tolerance	0.025 mm
Mesh quality	High
Total nodes	1333715
Total elements	943872
Maximum Aspect Ratio	11.172
Percentage of elements with Aspect Ratio < 3	99.9
Percentage of elements with Aspect Ratio > 10	0.000636
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:43
Computer name	

Figure16: mesh3(Fine mesh)

Mesh Details

Study name	fine mesh with load 9
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	0.5 mm
Tolerance	0.025 mm
Mesh quality	High
Total nodes	1333715
Total elements	943872
Maximum Aspect Ratio	11.172
Percentage of elements with Aspect Ratio < 3	99.9
Percentage of elements with Aspect Ratio > 10	0.000636
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:43
Computer name	

Figure 17: fine mesh with load 9N

Mesh Details

Study name	fine mesh with load 18
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	0.5 mm
Tolerance	0.025 mm
Mesh quality	High
Total nodes	1333715
Total elements	943872
Maximum Aspect Ratio	11.172
Percentage of elements with Aspect Ratio < 3	99.9
Percentage of elements with Aspect Ratio > 10	0.000636
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:43
Computer name	

Figuer:18 fine mesh with load 18N

3.2.3 Sample 10

This is the image of the sample 08, which is download from the Moodle and opened in the solid work software, which is shown in the figure 19.

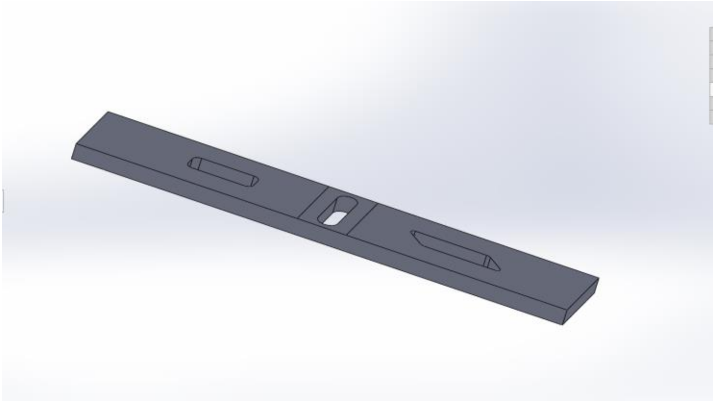


Figure 19: sample 10 model

Step 1: Applying the acrylic material into the sample08

- click in the simulation tab, then click on new study. Named it as a static 1. Then click ok.
- Right click in the sample 10 in the feature design tree and click on apply/edit material. Then select the acrylic material by clicking the plastic section. Then click on apply.

Step 2: Assigning the same boundary conditions to the sample08

- Here, the fixture is applied on either end, at a distance of 10mm from the flat surface as shown in the figure 20.
- Then, right click on the External load, apply the force (82.22 N)at the middle of 75mm from the either ends of the flat surface of the specimen. Which is opposite to the flat surface of the applied fixture as shown in the figure 21.

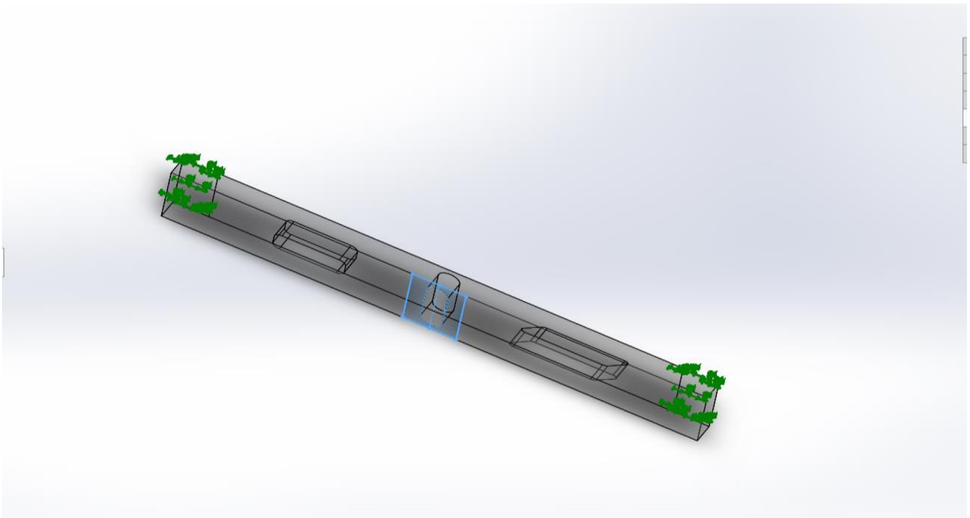


Figure 20: applying fixture in sample10

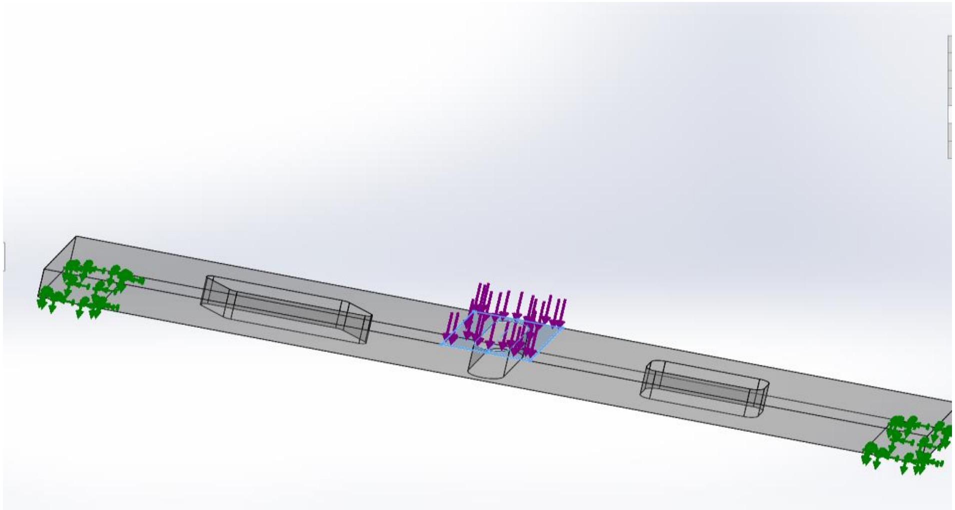


Figure 21: Applying load (82.22 N) in sample 10

Step 3: Applying the three different mesh

- There are three different types Mesh should run in three different studies with same boundary conditions by right clicking the static 1 and click the copy study and then assign the reimaging two mesh details. Which is shown in the figure below.

Mesh Details

Study name	Static 1 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	3.92852 mm
Tolerance	0.196426 mm
Mesh quality	High
Total nodes	5932
Total elements	3218
Maximum Aspect Ratio	9.7754
Percentage of elements with Aspect Ratio < 3	96.8
Percentage of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:01

(a)

Mesh Details

Study name	mesh1 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	2.57386 mm
Tolerance	0.128693 mm
Mesh quality	High
Total nodes	15259
Total elements	9067
Maximum Aspect Ratio	7.6288
Percentage of elements with Aspect Ratio < 3	99.4
Percentage of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:01

(b)

Mesh Details

Study name	mesh 2 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	1.35466 mm
Tolerance	0.0677332 mm
Mesh quality	High
Total nodes	78775
Total elements	51181
Maximum Aspect Ratio	5.366
Percentage of elements with Aspect Ratio < 3	99.8
Percentage of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:03

(c)

Figure 22: Types of mesh (Details)

Result:

4.1 Sample 05:

Table 2: sample 05 results

	Von Mises Stress (Mpa)		Resultant displacement mm		Equivalent Strain		Factor of safety	
	Max	Min	Max	Min	Max	Min	Max	Min
Static 1 (Mesh 1)	1.348e+01	4.505e-04	4.447e+00	1.00e-30	2.55e-03	1.878e-07	9.989e+04	3.339e+00
Mesh 2	3.072e+02	3.548e-03	7.361e+01	1.000e-30	6.607e-02	2.298e-06	1.268e+04	1.465e-01

- **Mesh 3(fine mesh)**

(a) Von Mises Stress

Maximum Von Mises stress- 6.146e+02 Mpa

Minimum Von Mises stress- 7.758e-04 Mpa

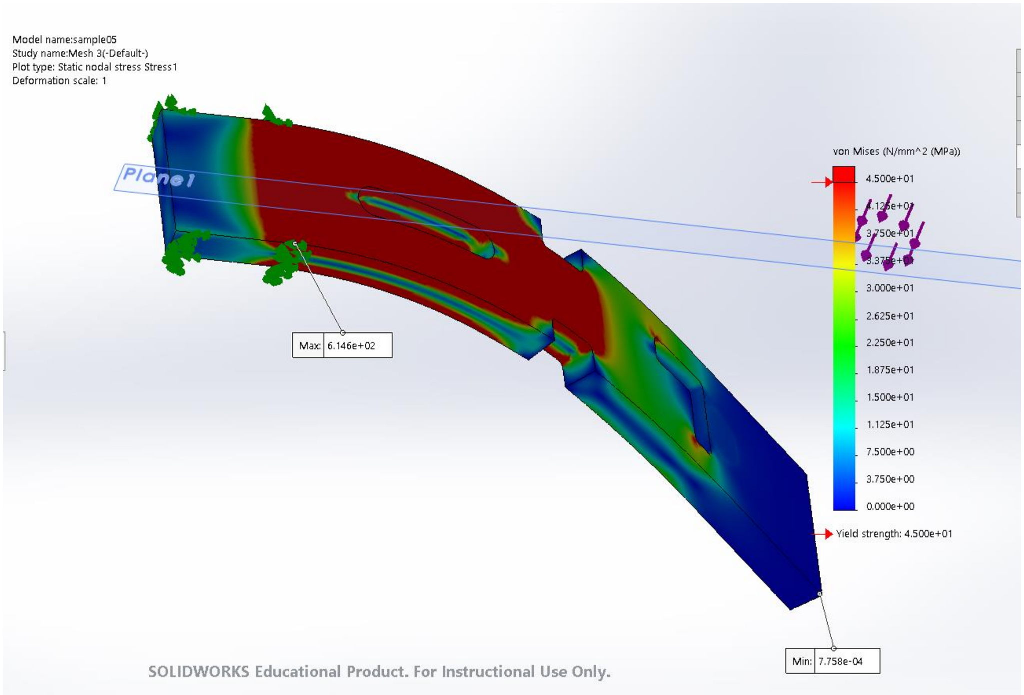


Figure 23: Von mises stress(sample 05)

(b) Resultant displacement

Maximum Resultant displacement - 7.389e+01 mm

Minimum Resultant displacement – 1.000e-30 mm

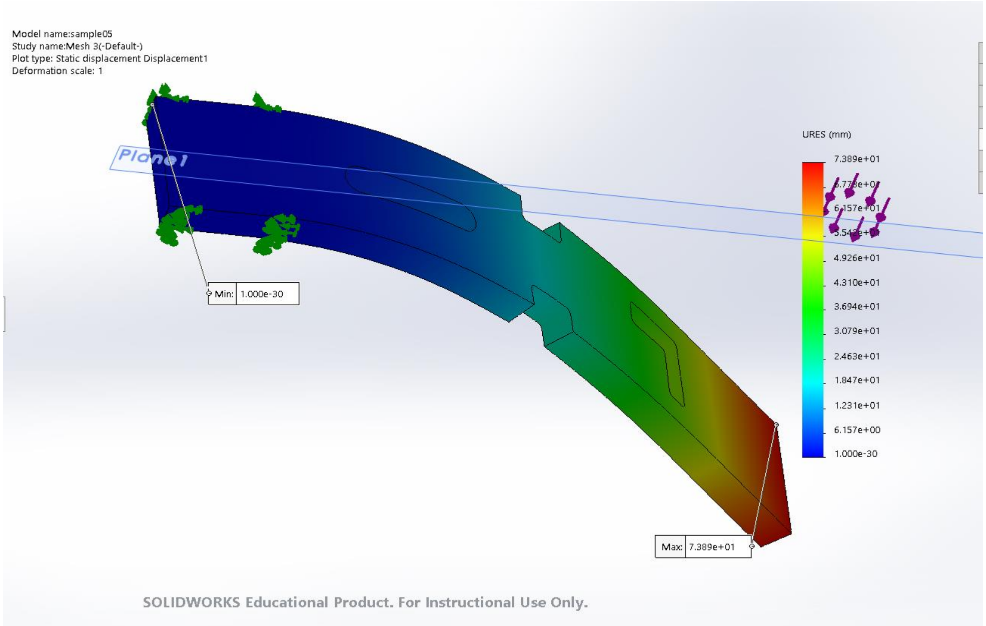


Figure 24: Resultant displacement

(c) Equivalent strain

Maximum Equivalent strain - 1.249e-01

Minimum Equivalent strain – 1.493e-07

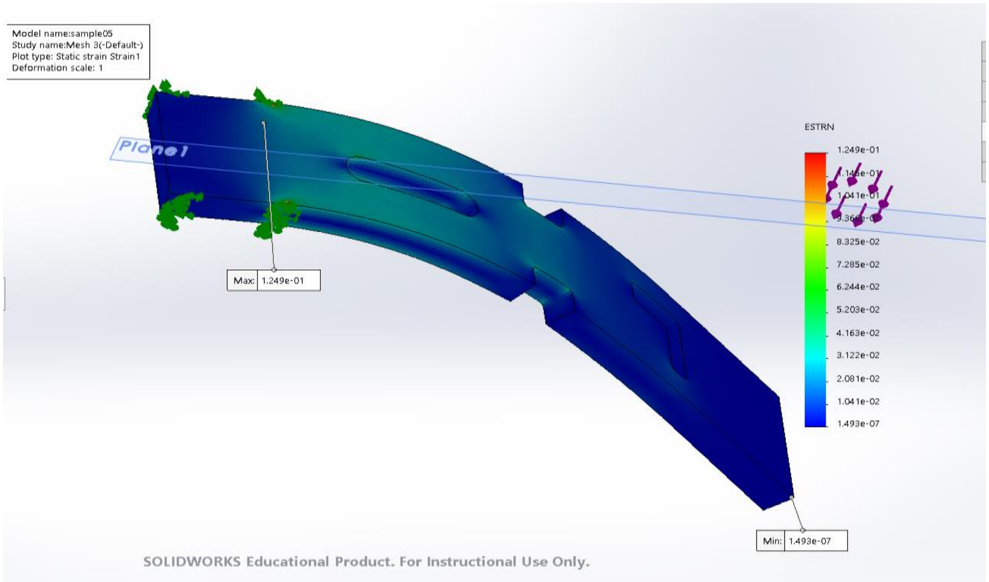


Figure 25: Equivalent strain (sample 05)

(d) Factor of safety

Maximum factor of safety – 5.801e+04

Minimum factor of safety - 7.322e-02

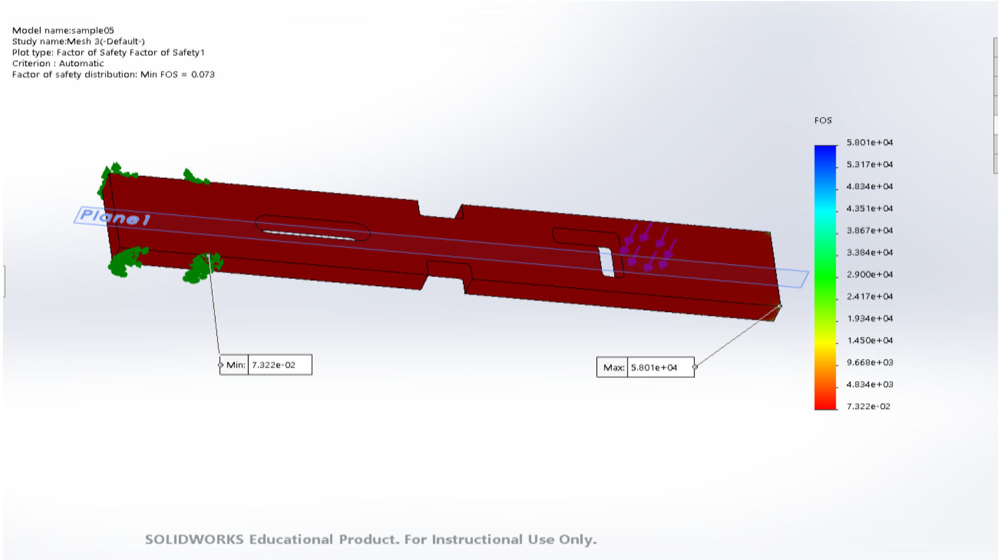


Figure 26: Factor pf safety (sample05)

4.2 Sample 08

Table 3: sample 08 results

	Von Mises Stress (Mpa)		Resultant Displacement (mm)		Equivalent strain		Factor of safety	
	Max	Min	Max	Min	Max	Min	Max	Min
Static 1 (Mesh 1)	6.718e+01	3.148e-04	1.794e+01	1.000e-30	1.387e-02	6.313e-07	3.000e+00	6.699e-01
Mesh 2	6.786e+01	1.919e-04	1.807e+01	1.000e-30	1.532e-02	1.504e-07	3.000e+00	6.631e-01
Fine mesh with load 9N	4.833e+01	2.476e-05	6.095e+00	1.000e-30	1.309e-02	1.165e-08	3.000e+00	9.312e-01
Fine mesh with load 18N	9.644e+01	5.108e-05	1.000e-30	1.217e+01	2.621e-02	2.323e-08	3.000e+00	4.666e-01

- **Mesh 3 (Fine mesh, load 27.05)**

(a) Von Mises Stress

Maximum Von Mises Stress -1.443e+02 Mpa

Minimum Von mises stress – 6.455e-05 Mpa

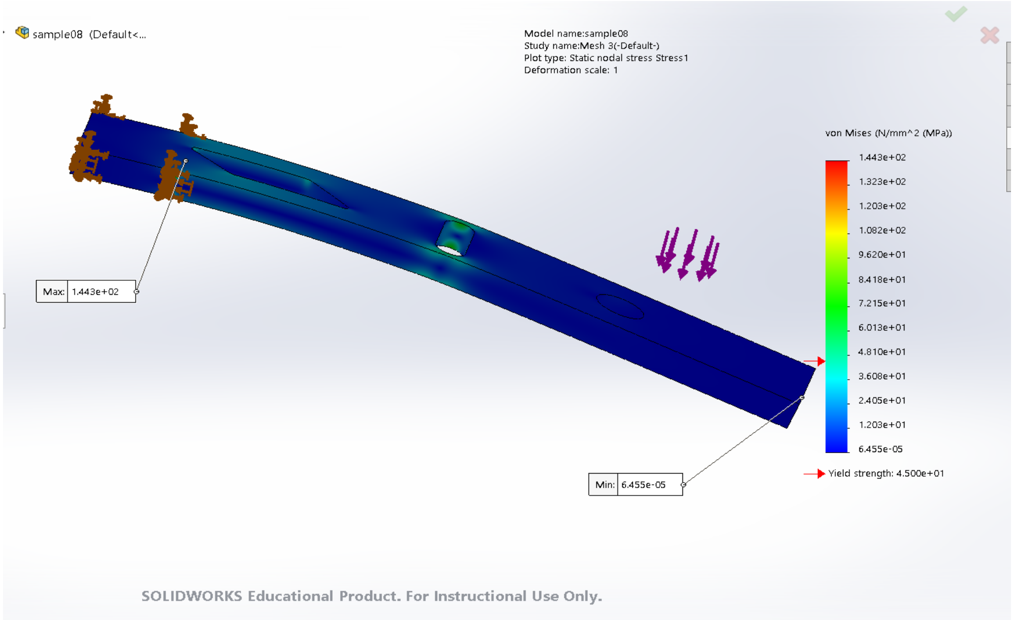


Figure 27: Von Mises Stress(sample 08)

(b) Resultant displacement

Maximum Resultant displacement – 1.821e+01 mm

Minimum Resultant Displacement – 1.000e-30 mm

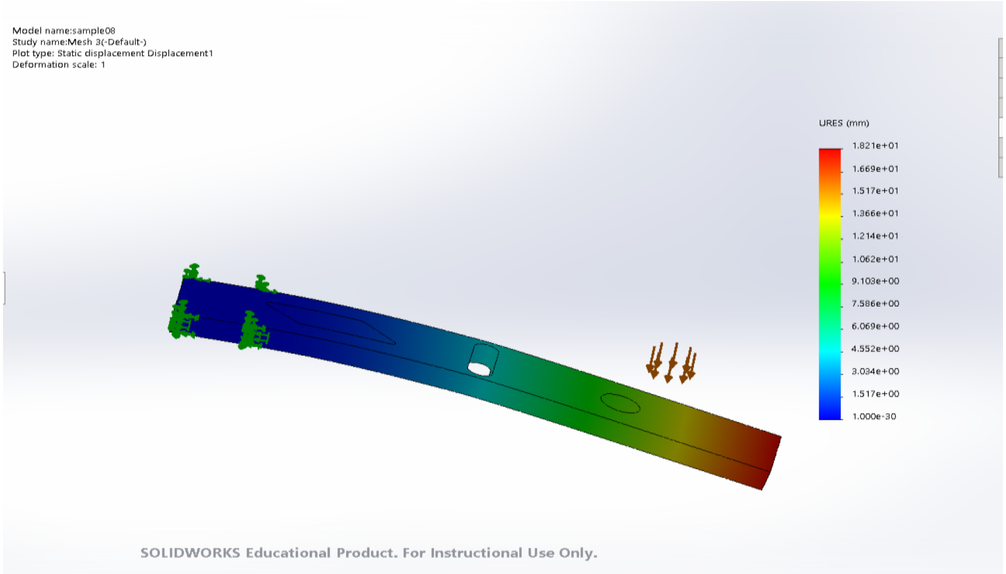


Figure 28: Resultant Displacement (sample 08)

(c) Equivalent strain

Maximum Equivalent strain – 3.935e-02

Minimum Equivalent Strain - 3.465-08

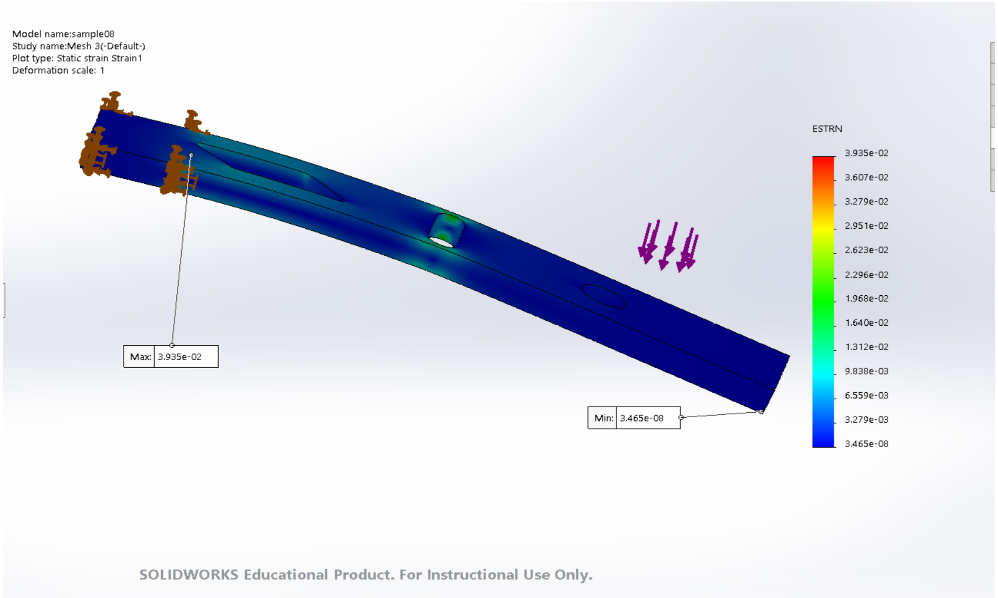


Figure 29: Equivalent strain (sample 08)

(d) Factor of safety

Maximum Factor of safety – 3.000e+00

Minimum Factor of safety – 3.118e-01

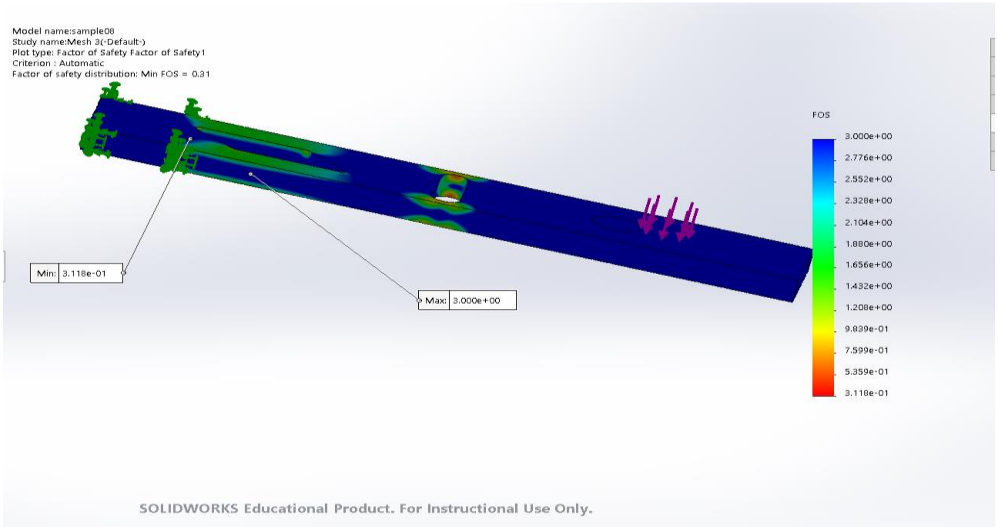


Figure 30: Factor of safety (sample 08)

4.3 Sample 10

Table 3: sample 10 results

	Von Mises Stress (Mpa)		Resultant displacement(mm)		Equivalent Strain		Factor of safety	
	Max	Min	Max	Min	Max	Min	Max	Min
(static 1) Mesh 1	5.191e+01	8.184e-02	1.271e+00	1.000e-30	8.092e-03	5.153e-05	5.499e+02	8.668e-01
Mesh 2	4.865e+01	6.278e-02	1.293e+00	1.000e-30	9.909e-03	1.063e-05	7.168e+02	9.249e-01

(a) Von Mises Stress

Maximum von mises stress – 5.282e+01 Mpa

Minimum Von mises stress – 3.199e-02 Mpa

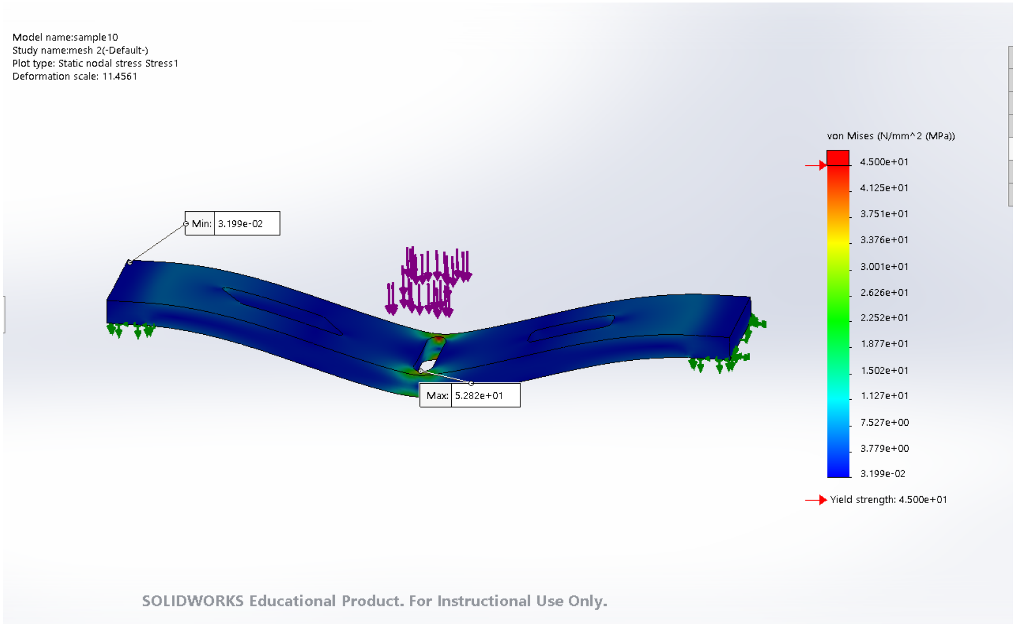


Figure 31: Von Mises stress (sample 10)

(b) Resultant displacement

Maximum resultant displacement - 1.309e+00 mm

Minimum Resultant Displacement- 1.000e-30 mm

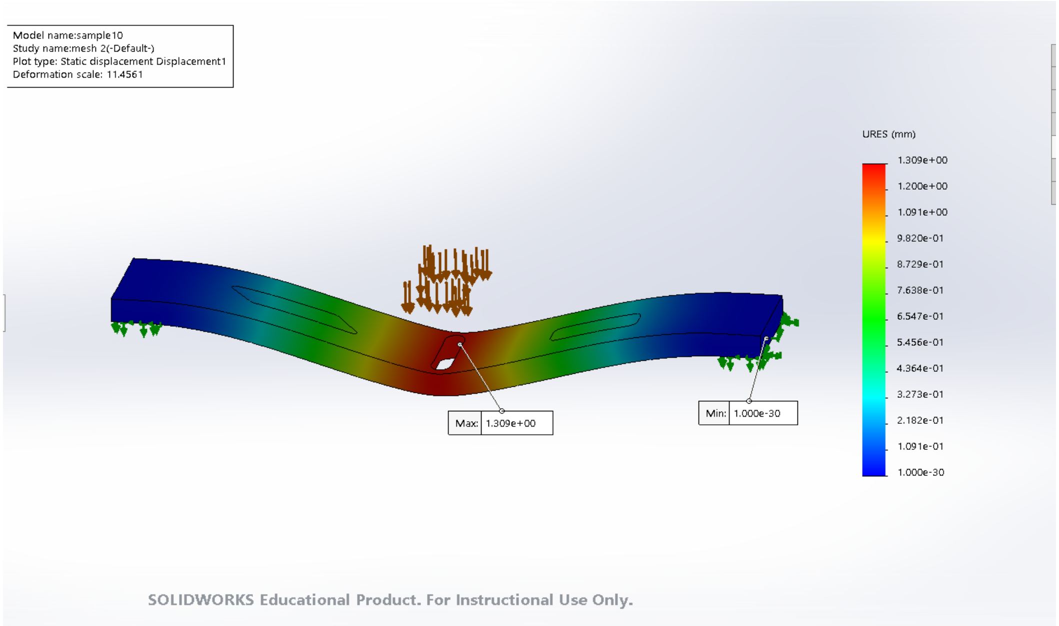


Figure 32: Resultant displacement (sample 10)

(c) Equivalent strain

Maximum equivalent strain- 1.263e-02

Minimum equivalent strain- 6.358e-06

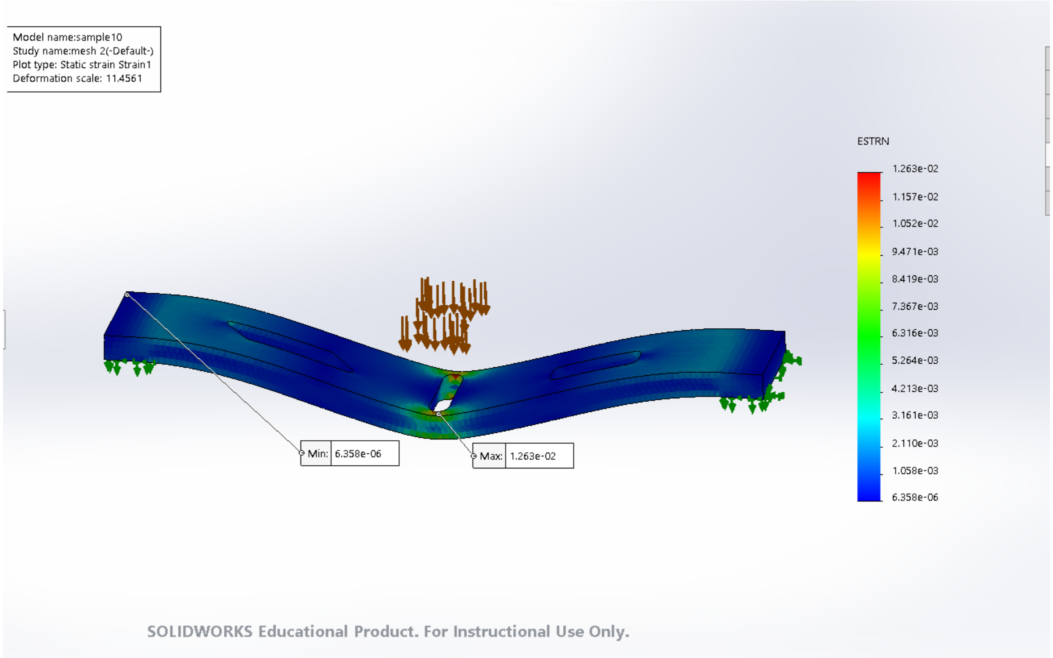


Figure 33: Equivalent strain (Sample 10)

(d) Factor of safety

Maximum Factor of safety – 1.406e+03

Minimum factor of safety – 8.520e-01

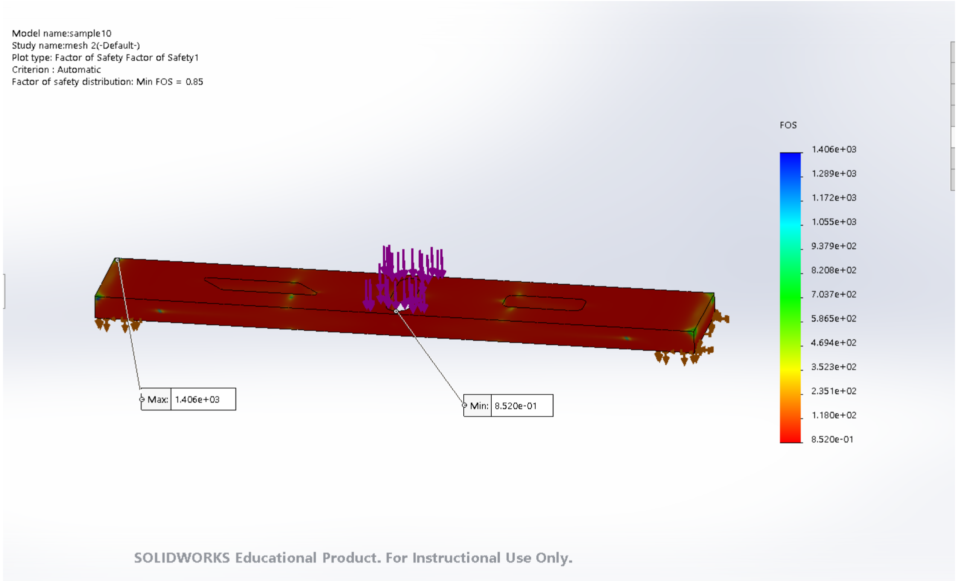


Figure 34: Factor of safety (sample 10)

5.0 Discussion:

1. Sample 05

- In the case of the sample 05, if we go to the result of the von mises stress. The maximum yield strength of the acrylic material is 4.500e+01 Mpa. However, the maximum von mises stress is near the fixture shown in the figure 23. Hence, the von mises stress is greater than the yield strength. Consequently, the percentage of breaking is high in that area compared to other area.

- In the laboratory method the material does not breaks. Because, there might have some chance of having errors due to area of load applied, errors in the equipment.

2. Sample 08

- The percentage of breaking is high at the centre of the acrylic sample 08 in the Finite element analysis. The reason for the failure of the same 08 is the geometric profile is at the centre of the acrylic sample. So, when an external load of 27.05 N is applied at the flat surface as shown in the figure 36. In the centre the von mises stress (5.0002e+01(Mpa)) exceeds the yield strength(4.500e+01(Mpa)). Hence the failure occurs in the middle.

- In the laboratory method, the location of failure is almost like the finite element analysis. So, here in this laboratory method the reason for the failure at the centre is due to the cut-out section at the centre. So, the surface area at the centre is less. Hence the sample 08 fail.

3. **Sample 10**
   - Sample 10 failures at the centre of the specimen, because the cross-section area at centre is less. So, the stress is more compared to other areas of this specimen. Similarly, value of von mises is greater than the yield strength. Hence, the percentage of failure is at the centre as shown in the figure 31.

   - Position of the failure of sample 10 in the laboratory is at the centre, which is almost like the Finite element analysis of sample 10. Here also the reason for the failure is the geometric profile.

Laboratory method:

- In the laboratory method, if there is any chances in the position of the load applied. So, errors can occur.
- If there are any minute errors in the equipment's which is used for this experiment might leads to errors in the values. Therefore, by using accurate equipment's leads a good result without having any errors.

Finite Element Analysis:

- To get an accurate result, we need to give fine mesh by reducing the size of the mesh. Then we will get good result with less errors.
- Second reason for getting finite result is the boundary conditions. Because, if we gave wrong boundary conditions the results will change.
- Third reason is that, in the solid works the properties of the materials is already defined, Hence the values are constant. But in the real world conditions the properties of the material might change hence we need to edit the properties of the material to get a good result.

5.0 Reference:

- https://learning.wintec.ac.nz/mod/resource/view.php?id=1327678&redirect=1
- https://learning.wintec.ac.nz/mod/folder/view.php?id=1327684

- https://learning.wintec.ac.nz/course/view.php?id=11736&section=2