Master Thesis

NTNU
Norwegian University of Science and Technology
Faculty of Engineering
Department of Mechanical and Industrial Engineering

Morten Mosvold Jørgensen

Design of nature-inspired components with tailored geometry for high mechanical performance

TMM4960 – Product Development and Materials, Master Thesis

Trondheim, June 2021

NTNU
Norwegian University of
Science and Technology

Design of nature-inspired components with tailored geometry for high mechanical performance

"Biomimetic has begun to offer significant potential in providing unique solutions in materials science. However, the full potential of mimicking natural systems can be fully exploited if we start looking at the problem of design of complex structures from the structural side. This project focuses on investigating simple case studies to show the possibility of achieving an ideal design with minimum stress raisers inspired by nature. Controlling the stress condition in the structure would eventually lead to better mechanical performance under static and cyclic loading conditions. The student will be working on CAD software, available software packages for the design of lattice structures, finite element analysis of the design, 3D printing in the lab and, last but not least, mechanical testing of the designed structures."

Preface

This thesis is written as a product of the master thesis work carried out by the Department of Mechanical Engineering and Production at the Norwegian University of Science and Technology during the fall of 2020 and the spring of 2021. This thesis serves as a contribution to the work of Javad Razavi, as well as aims to increase the knowledge of nature-inspired geometry in general.

The work has been rewarding, and it has given me the opportunity to study the complex field of NID alongside far more experienced individuals.

I would like to thank Mr. Razavi for the opportunity to work on this project, and for his contribution to the thesis. I would not be able to do it without his expertise, good will, support and guidance. I would also like to thank Mr. Zhuo Xu for his good contribution for the thesis during the work carried out in the spring.

Special thanks to the staff in the laboratory, who was able to host safety courses, produce test specimens and help with general questions from a student with mostly theoretical previous experience.

Abstract

The purpose of this thesis is to study the effect of adding tailored geometries in specimens to strengthen them and increase their fatigue resistance. Several studies are performed on different geometries. The specimens are produced from aluminum, and water-cut into the desired shape. The results from the experiments show that the fatigue life of the best performing specimens more than double the fatigue life over the control-specimens, simply by cutting holes in the specimen above and below the center hole. The results show that the added voids play an important role in increasing the fatigue life by redistributing the stress and better utilizing the lesser-stressed parts of the specimens.

Table of Contents

Table of Figures

Table of Tables

Nomenclature

NID	Nature-Inspired Design
TO	Topology Optimization
FE	Finite Element
FEA	Finite Element Analysis
BC	Boundary Constraint
Design Variables	A numerical input value that is allowed to Change throughout the optimization.
E	Modulus of elasticity
Ntot	Total fatigue life
W	Width of cracked specimen
ν	Poisson´s ratio
σ	Applied stress to specimen
ASTM	American Society for Testing and Materials
AM	Additive Manufacturing
Lattice Structure	A space-filling unit cell

Introduction

The thesis is suited for anyone with novice to advanced knowledge about nature-inspired design, who wants to better understand the impact of tailoring geometries for a better stress redistribution and utilize lesser stressed areas of a part.

Javad Razavi presented the subject through the master thesis task publication. The applications of nature-inspired design appealed; their popularity was certain to increase with the advancements in additive manufacturing and computational mechanics. The study of topology-optimized nature-inspired design is well researched. However, the study on tailoring the geometry for fatigue optimization on said topics is not as well researched and deserves more attention.

The concept of removing material as a means to increase strength is something nature has been doing through trial and error; however, traditional engineering practice has not focused on this. Throughout this master thesis, we design and tweak certain design variables to try to get closer to an ideal design of a dogbone with a center hole. The master thesis is a continuation of work carried out in the fall of 2020, and thus, adds more complex designs, as well as re-tests several geometries that were flawed in production from the fall.

One of the main failure mechanisms in engineering components and structures is fatigue failure. Most materials seem to experience a fatigue-related failure [1]. Hence, the investigation continues for measures to increase the life of structures. We analyze various methods aimed at increasing the strength of structures, as well as one believed to increase strength that does the opposite.

In 1986, the paper "Changes in the Fatigue Life of Plates with Attachments Due to Geometrical Effects" was published. [2] I. F. C. Smith and T. R. Gurney investigated how a metal plate with attachments welded on top (a) would have a shorter fatigue life than one without (b). Thus, one might conclude that decreasing the thickness of the plate like in (c) would increase fatigue life, but that is obviously also not the case. If specimens (a), (b) and (c) would be stretched a fixed length under a fatigue test until failure, specimen (b) would experience the longest fatigue life, while specimen (a) would fail earlier, since the thin part would endure most of the stretching while the thick part would be stiff.

Figure 1: Specimen (a) under tension.

Figure 2: Specimen (b) under tension.

Figure 3: Specimen (c) under tension.

In addition to varying thickness, changing the geometry surrounding sensitive areas would impact fatigue life. For figures 4 and 5, Kt was reduced from 1.43 for (e) to 1.07 for (d). [3]

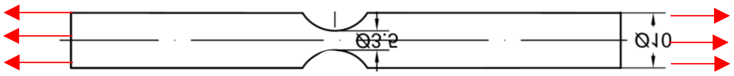


Figure 4: Geometry (d)

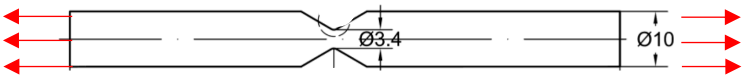


Figure 5: Geometry (e)

High toughness, high strength and resistance to fatigue failure are basic requirements for most engineering constructions. Studies have shown how the weight can be drastically reduced, while maintaining the strength. A good example is how Bugatti reduced the weight by 40% of their brake calipers, while increasing the strength by 20%. [4] Several parts of the brake caliper resemble how skeletons would grow in humans to build as strong structures as possible when there is no limitation in the manufacturing process.

Managing stress by manipulating where the load travels through the specimen is crucial for long life and lighter weight. It is not only about where to put the material; it is about where not to put the material. We look into how removing material and tailoring the voids will force the stress into lesser utilized parts of the specimen. Stress and fatigue life throughout the FEA and experimental tailoring are obtained and used as a basis for discussing how various design variables affect the specimens.

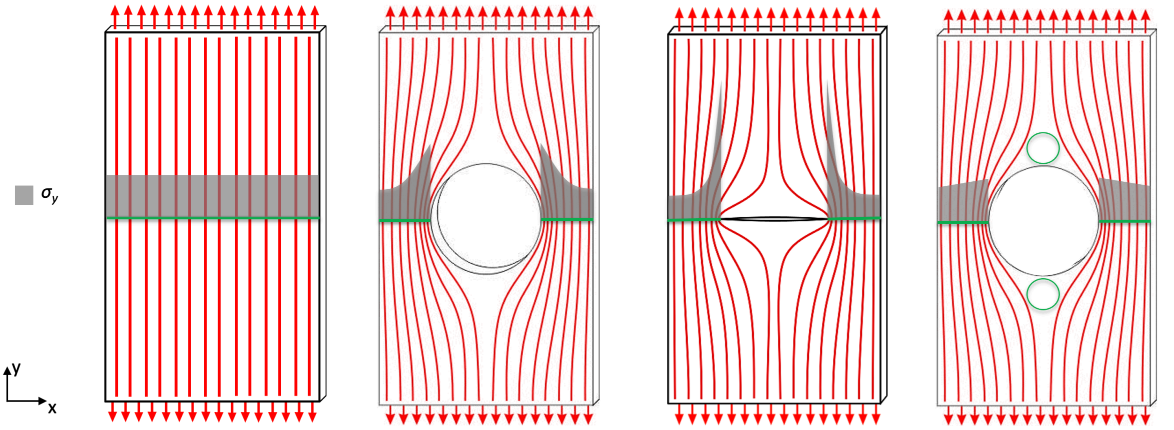


Plate under tension
Circular notch
crack
Circular notch with added tailored notches

Figure 6: Demonstrating how tailored voids affect stress in the specimen [5]

Figure 6 shows how the stress travels through specimens under tensile load, with various geometries affecting the stress path. The specimen to the far right has two added circular voids that “push” the stress into the lesser stressed parts. Throughout the thesis, we vary certain design variables, with the goal of increasing fatigue life through better stress redistribution.

The thesis work carried out is divided into:

Done during fall semester:

- Initial FEA playing around with several geometries and design variables.
- FEA of dogbones modelled after the ASTM standard with inspiration from the initial FEA.
- Fatigue test of the dogbones.

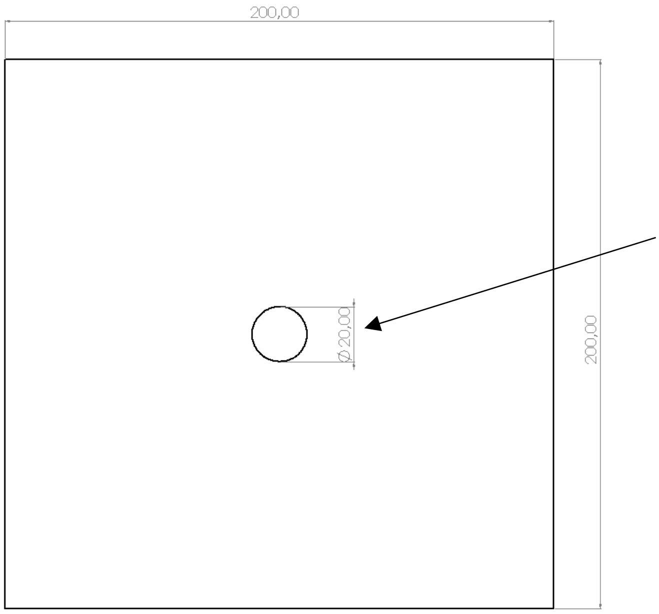
Done during spring semester:

- New FEA with tweaked geometries.
- Fatigue test of new geometries.
- Final geometries with added lattice structures produced with nTopology.

Initial FEA

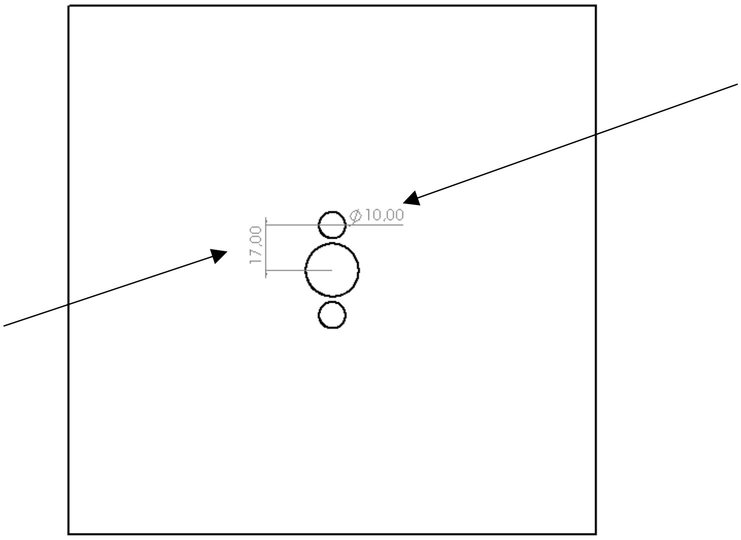
Geometries

A selection of 5 geometries was proposed for a stress analysis. Case 1 is a 200 x 200 square with a center hole of r=10. Cases 2-5 add four different geometries above and below the center hole. For each geometry, the radius and distances will be varied, all with the intention of manipulating the stresses in the material into a better distribution and increasing the strength and fatigue resistance. We want to avoid concentrations of stress in the material close to the center hole. The stresses in the center hole were obtained. Throughout the report, the cases are referred to with their respective case numbers. The center hole was fixed at 10mm all the time. The following are the five geometries shown.



Radius of center hole fixed at all times in all cases

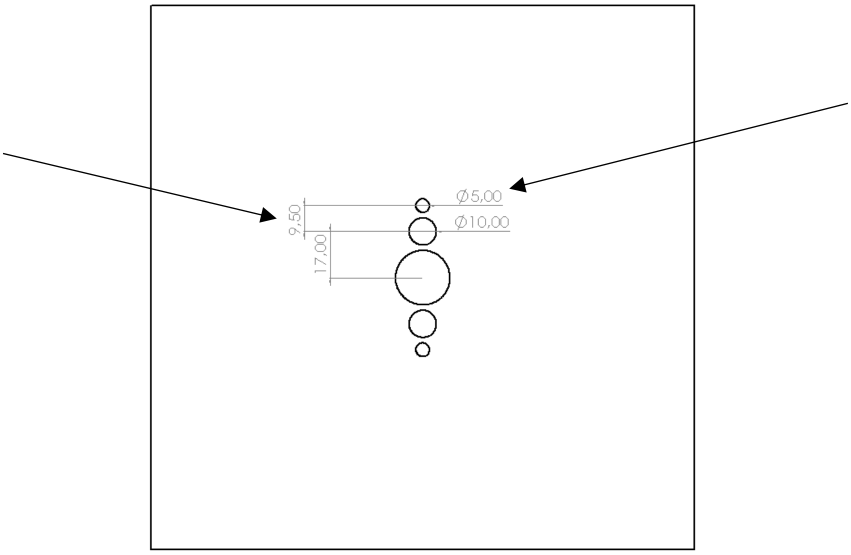
Figure 7: Case 1



Radius 1= Radius of support hole 1. It varied between 5, 7.5 and 10 mm

v1=Distance between center hole and support hole. It varied between 2, 3 and 4 mm (17-10-5=2mm)

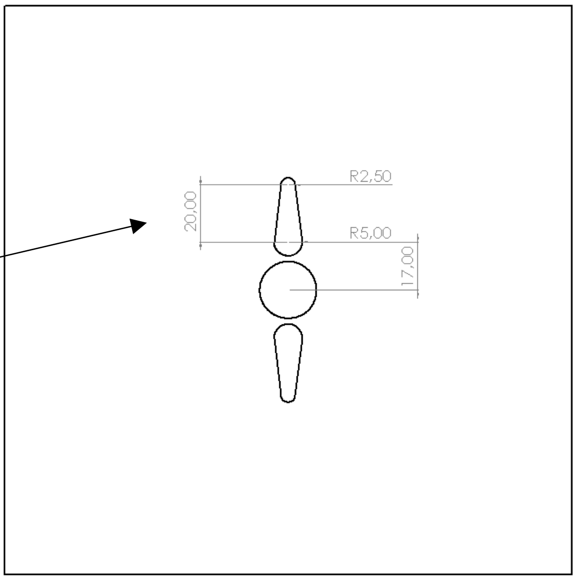
Figure 8: Case 2



v2=Distance between support hole 1 and support hole 2 . It varied between 2, 3 and 4 mm (9.5-5-2.5=2mm)

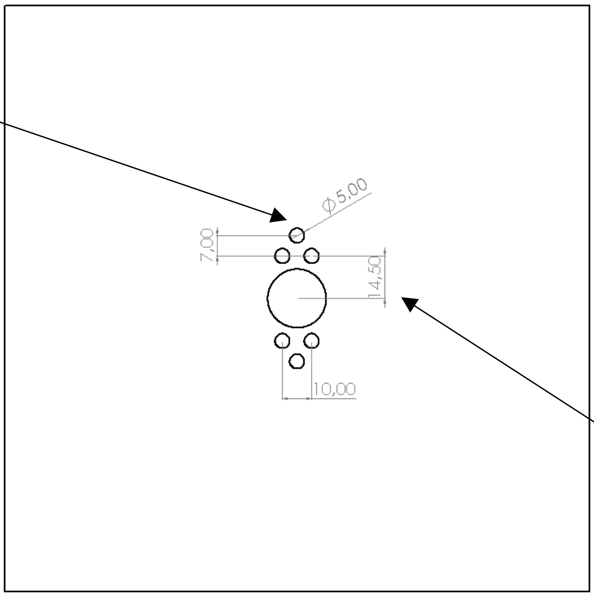
Radius 2= Radius of support hole 2. It varied between 2.5, 5 and 7.5 mm

Figure 9: Case 3



Same hole variations
on Case 4
as on case 3, except
for the distance
between
support hole 1 and 2,
which is fixed at
20 mm the whole
study.

Figure 10: Case 4



Radius of support
holes varied
between 2.5,
5 mm

Distance in Y-direction
between each set of holes
varied between
2, 3 and 4 mm
(14.5-10-2.5=2mm)

Figure 11: Case 5

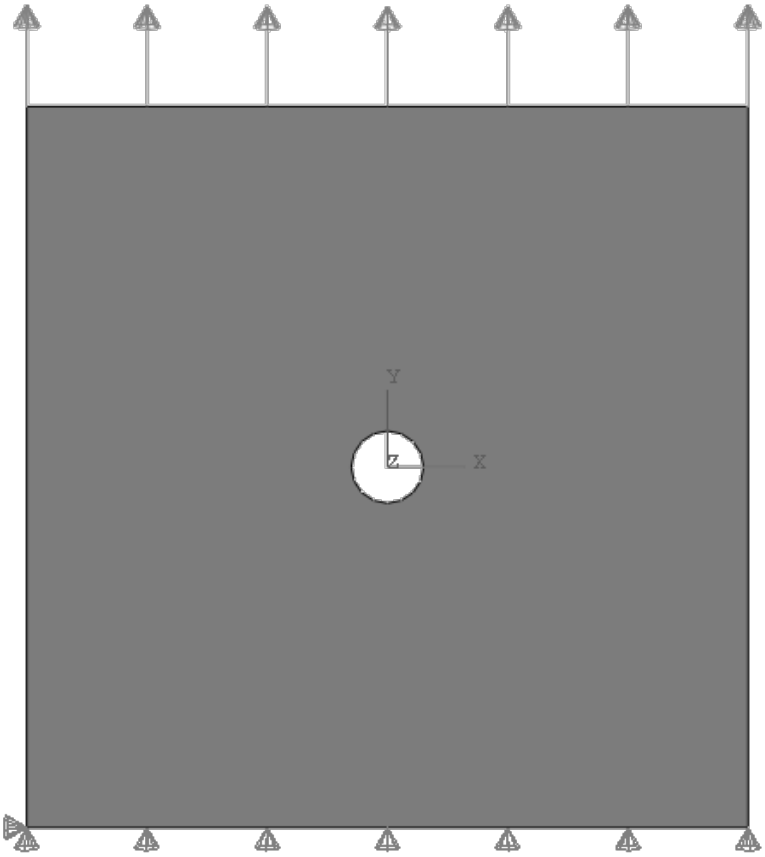


Figure 12: BC & Loads

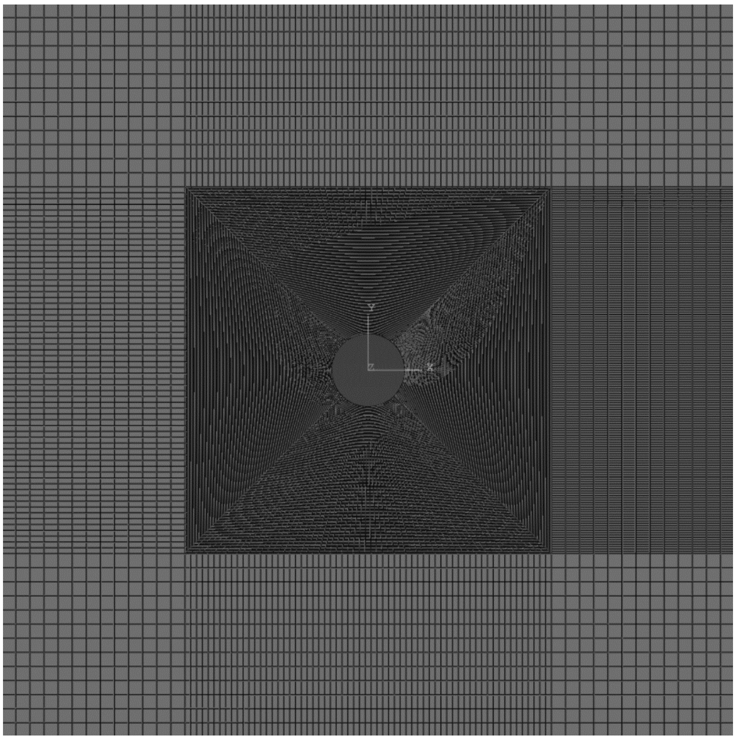


Figure 13: Mesh

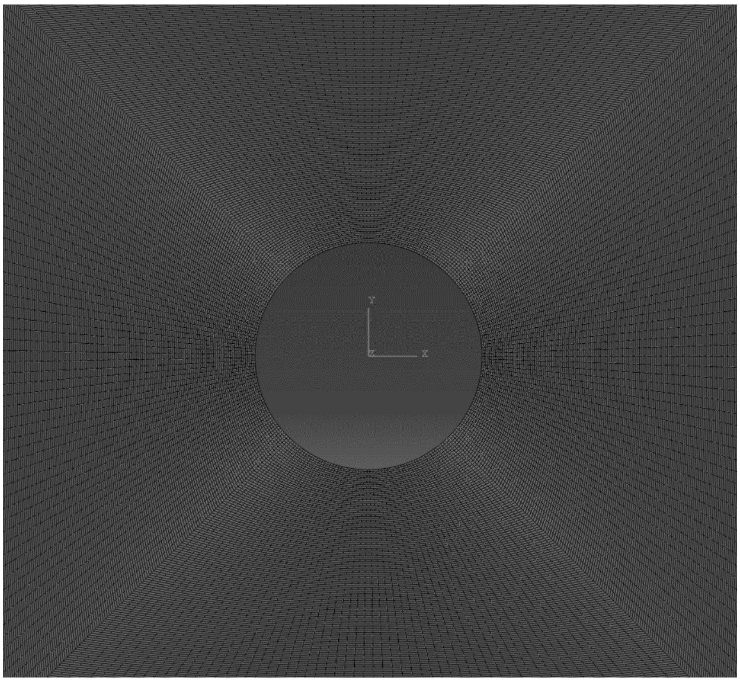


Figure 14: Mesh, detailed view

Detailed FEA procedures

The FEA test was performed with Abaqus 2017 software. The steps taken during the analysis are provided to be able to replicate the study in the future.

Material properties & load:
E = 70 000
ν = 0.33
σ = 100 MPa

1. Start Abaqus:
Start – Dassault Systems SIMULIA Abaqus CAE 2017 – Abaqus CAE.

2. 2. Create file name:
   Menu: File, Save as, New directory [Project], Select new directory, [Project], OK

3. Create Geometry:
Module: Part
   Menu: Part, Create,
   Name [Part-1], 2D, Deformable, Shell, Approx. size [200], Cont.
   # sketch mode is now active
Menu: Create Lines, Rectangle,
   Create rectangle with 200x200 dimensions
Menu: Add, Circle, Center/Perimeter,
   # pick center and radius of 10
Further geometries will be created in the same manner, and they will be specific to each part. Geometries are shown in detail in figures.

4. Create material and section properties.

Module: Property
    Menu: Material, Create, Name [Material-1], Mechanical, Elasticity, Elastic, Type, Isotropic [70000, 0.33], OK
Menu: Section, Create, Name [Section-1], Solid, Homogeneous, Cont.
    Material: OK
Menu: Assign, Section
    # pick all, Done
    OK

5. Create assembly

Module: Assembly
Menu: Instance, Create, Independent, OK

6. Create steps

Module: Step
Menu: Step, Create, Name [Step-1], General, Static/General, Cont.
OK

7. Create loads and boundary conditions

Module: Load
Menu: Load, Create, Name [load-1], Pressure, Cont.
    # pick the horizontal top edge on the top, Done, Magnitude [-100], OK
Module: Load
Menu: BC, Create, Name [BC-1], Step: Initial, Mechanical, Displacement/Rot, Cont.
    # pick horizontal bottom edge, Done; # Checkmark U2, OK
Module: Load
Menu: BC, Create, Name [BC-2], Step: Initial, Mechanical, Displacement/Rot, Cont.
    # pick bottom left corner, Done; # Checkmark U1, OK

8. Create meshing

Module: Mesh
Menu: Seed, Instance, Approx. global size [4], Apply, OK
Menu: Seed, Edges, Local Seeds, Select edge around center circle, Approx. element size [0.25], Apply, OK. Mesh size was refined until converging began.
Same step for all circles added later to the other geometries

9. Submit job

Module: Job
Menu: Job, Manager, Create, Name [Job-1], Model: [Model-1], Cont., OK, Submit

Results

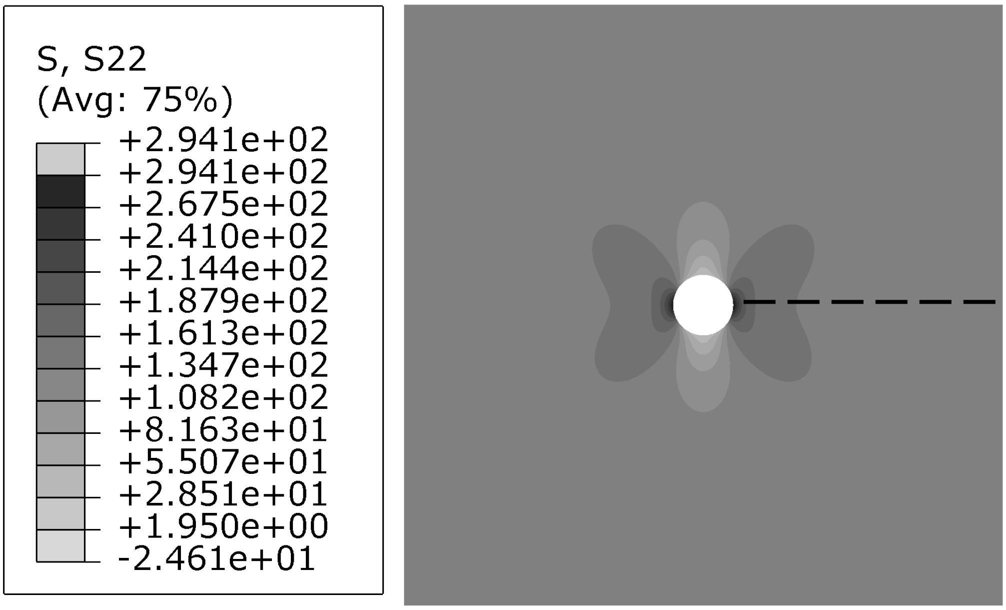
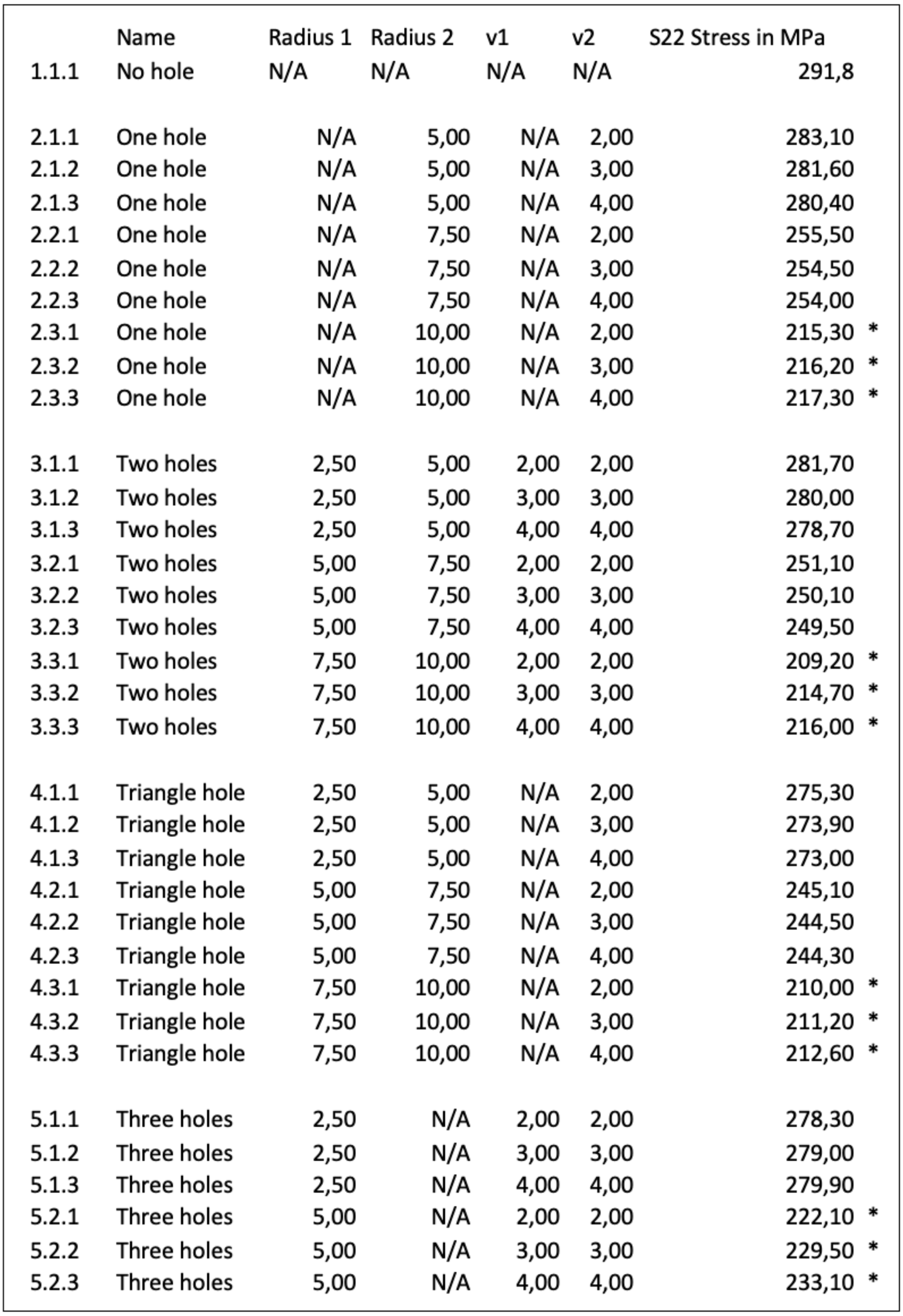


Figure 15: Example of Max S22 stress and stresses along path

The dotted line is referred to as “True distance along path”, the same name as Abaqus assigns it. X-value of zero is at the center hole, x-value of 90 is at the right edge. The stress along the line is obtained and presented for each case. The maximum occurring stress for all cases is situated closest to the center, and will also be presented in a separate diagram in Table 1. The design variables for each case is “Radius 1”, “Radius 2”, “v1” and “v2”.
Contour limit on legends for the figures is set to the maximum occurring stress, which happens at Case 1.1.1 at 294.1 MPa for the S22 stress. Minimum limit is set to -24.609 MPa which is the lowest occurring stress, happens at Case 3.1.1.



	Name	Radius 1	Radius 2	v1	v2	S22 Stress in MPa	
1.1.1	No hole	N/A	N/A	N/A	N/A	291,8	
2.1.1	One hole	N/A	5,00	N/A	2,00	283,10	
2.1.2	One hole	N/A	5,00	N/A	3,00	281,60	
2.1.3	One hole	N/A	5,00	N/A	4,00	280,40	
2.2.1	One hole	N/A	7,50	N/A	2,00	255,50	
2.2.2	One hole	N/A	7,50	N/A	3,00	254,50	
2.2.3	One hole	N/A	7,50	N/A	4,00	254,00	
2.3.1	One hole	N/A	10,00	N/A	2,00	215,30	*
2.3.2	One hole	N/A	10,00	N/A	3,00	216,20	*
2.3.3	One hole	N/A	10,00	N/A	4,00	217,30	*
3.1.1	Two holes	2,50	5,00	2,00	2,00	281,70	
3.1.2	Two holes	2,50	5,00	3,00	3,00	280,00	
3.1.3	Two holes	2,50	5,00	4,00	4,00	278,70	
3.2.1	Two holes	5,00	7,50	2,00	2,00	251,10	
3.2.2	Two holes	5,00	7,50	3,00	3,00	250,10	
3.2.3	Two holes	5,00	7,50	4,00	4,00	249,50	
3.3.1	Two holes	7,50	10,00	2,00	2,00	209,20	*
3.3.2	Two holes	7,50	10,00	3,00	3,00	214,70	*
3.3.3	Two holes	7,50	10,00	4,00	4,00	216,00	*
4.1.1	Triangle hole	2,50	5,00	N/A	2,00	275,30	
4.1.2	Triangle hole	2,50	5,00	N/A	3,00	273,90	
4.1.3	Triangle hole	2,50	5,00	N/A	4,00	273,00	
4.2.1	Triangle hole	5,00	7,50	N/A	2,00	245,10	
4.2.2	Triangle hole	5,00	7,50	N/A	3,00	244,50	
4.2.3	Triangle hole	5,00	7,50	N/A	4,00	244,30	
4.3.1	Triangle hole	7,50	10,00	N/A	2,00	210,00	*
4.3.2	Triangle hole	7,50	10,00	N/A	3,00	211,20	*
4.3.3	Triangle hole	7,50	10,00	N/A	4,00	212,60	*
5.1.1	Three holes	2,50	N/A	2,00	2,00	278,30	
5.1.2	Three holes	2,50	N/A	3,00	3,00	279,00	
5.1.3	Three holes	2,50	N/A	4,00	4,00	279,90	
5.2.1	Three holes	5,00	N/A	2,00	2,00	222,10	*
5.2.2	Three holes	5,00	N/A	3,00	3,00	229,50	*
5.2.3	Three holes	5,00	N/A	4,00	4,00	233,10	*

*There is now an additional higher maximum stress occurring in the support hole.

Tabell 1: Max S22 Stress at center hole

Tabell 2: Max S22 Stress at center hole

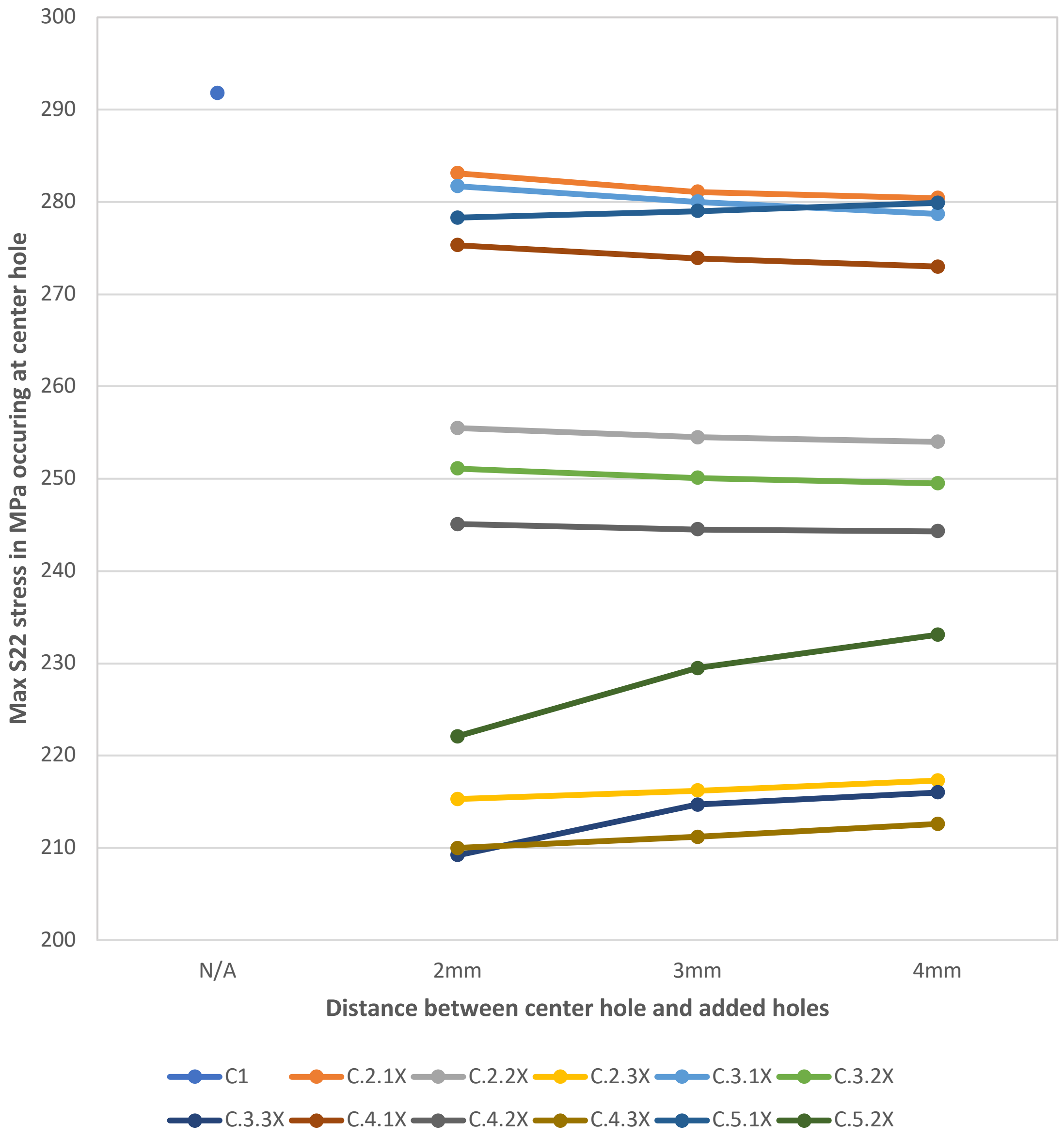


Figure 16: Maximum stresses at center hole for all cases plotted along distance variation on x-axis.

The Max S22 stress from the table is plotted along the distance variation. Each line represents a different hole size. The last number for the case name in Figure 10 is replaced with "X" since it varies with a distance of 2, 3 and 4 mm.

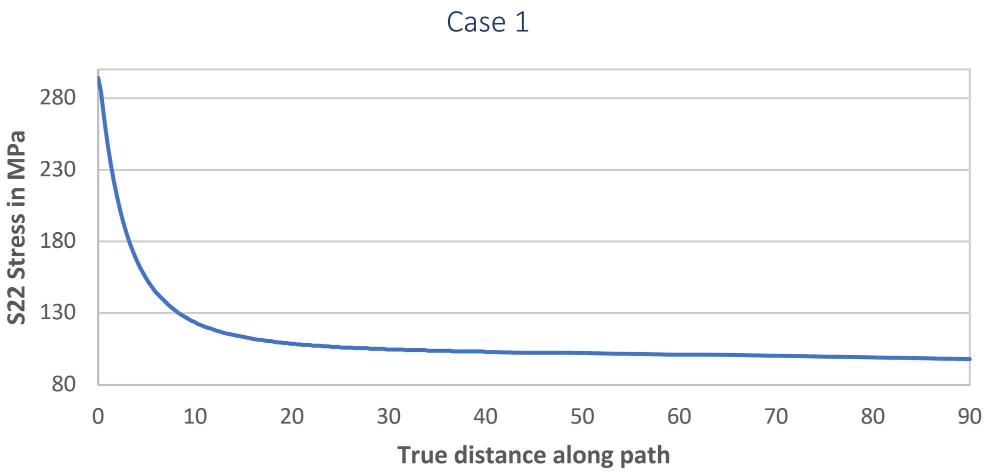


Figure 17: Stress along path for Case 1

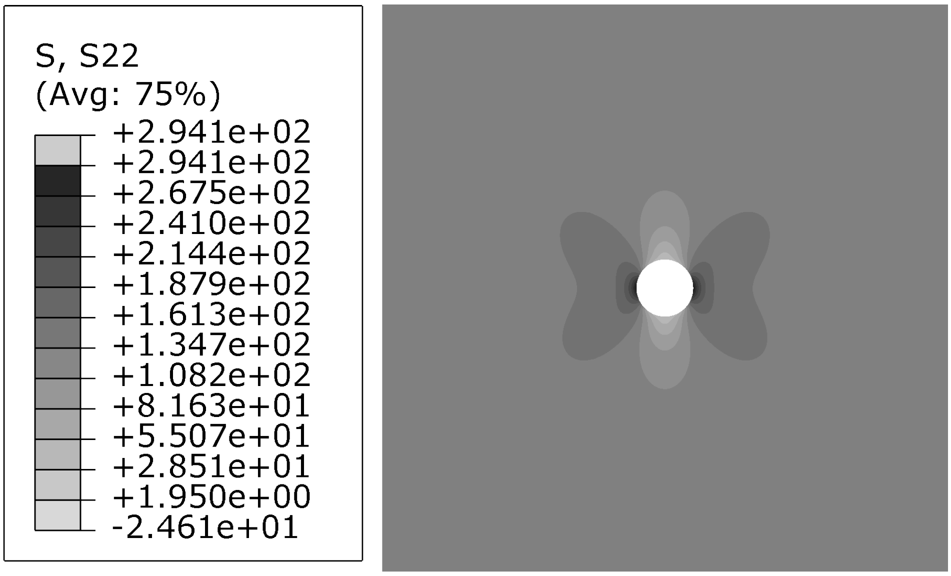


Figure 18: Stress contours for Case 1.1.1

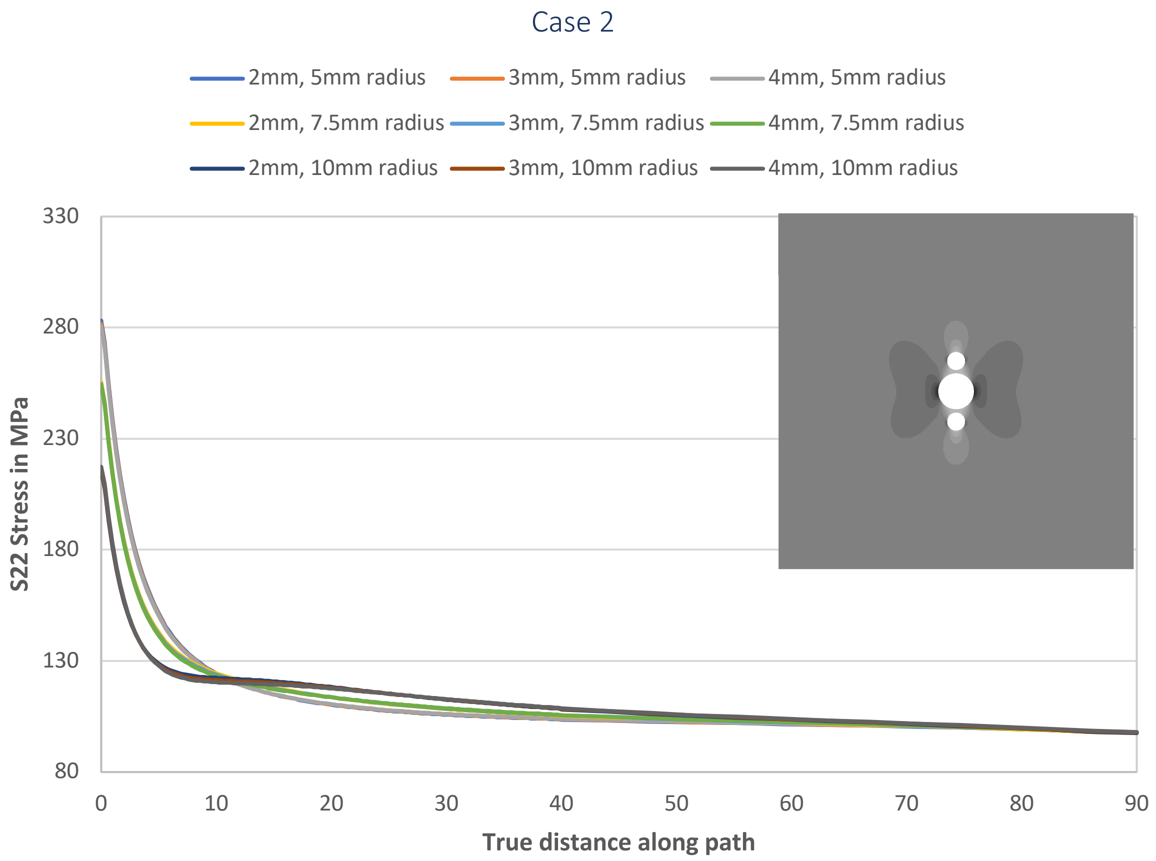


Figure 19: Stress along path for Case 2

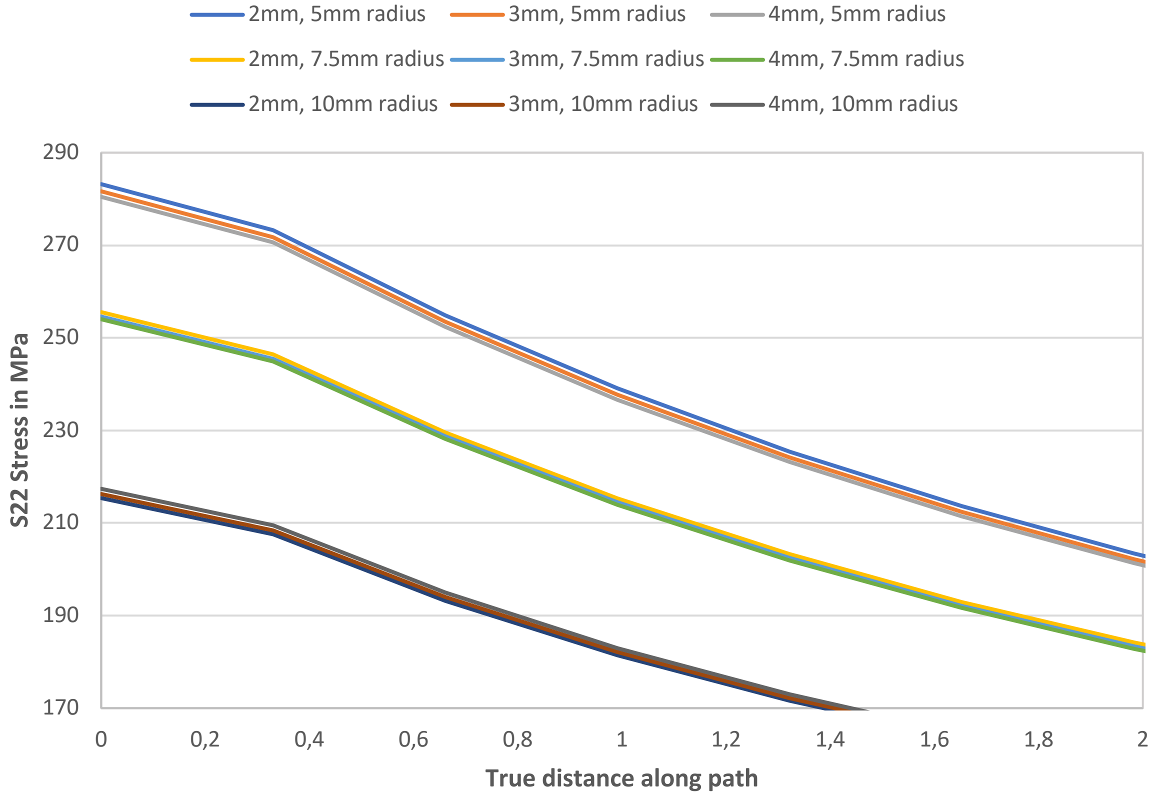


Figure 20: Magnified view of stress along path for Case 2 for section 0 to 2

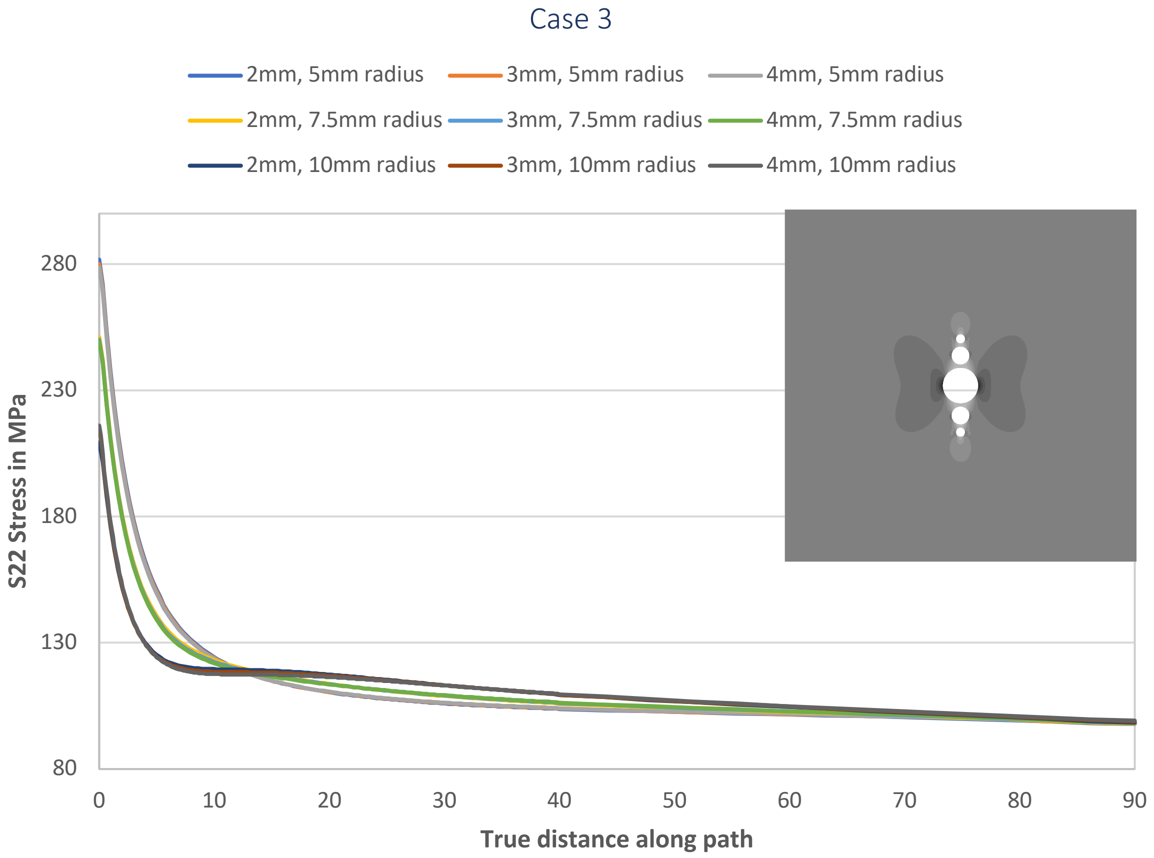


Figure 21: Stress along path for Case 3

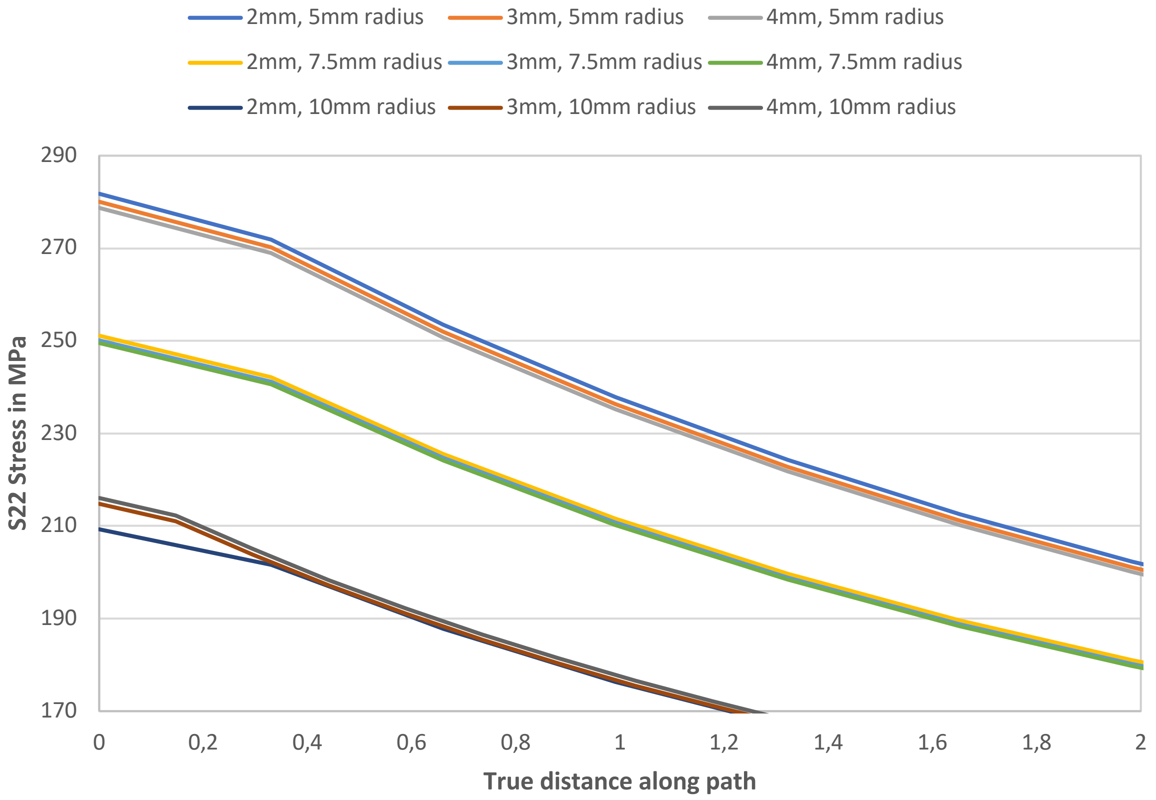


Figure 22: Magnified view of stress along path for Case 3 for section 0 to 2

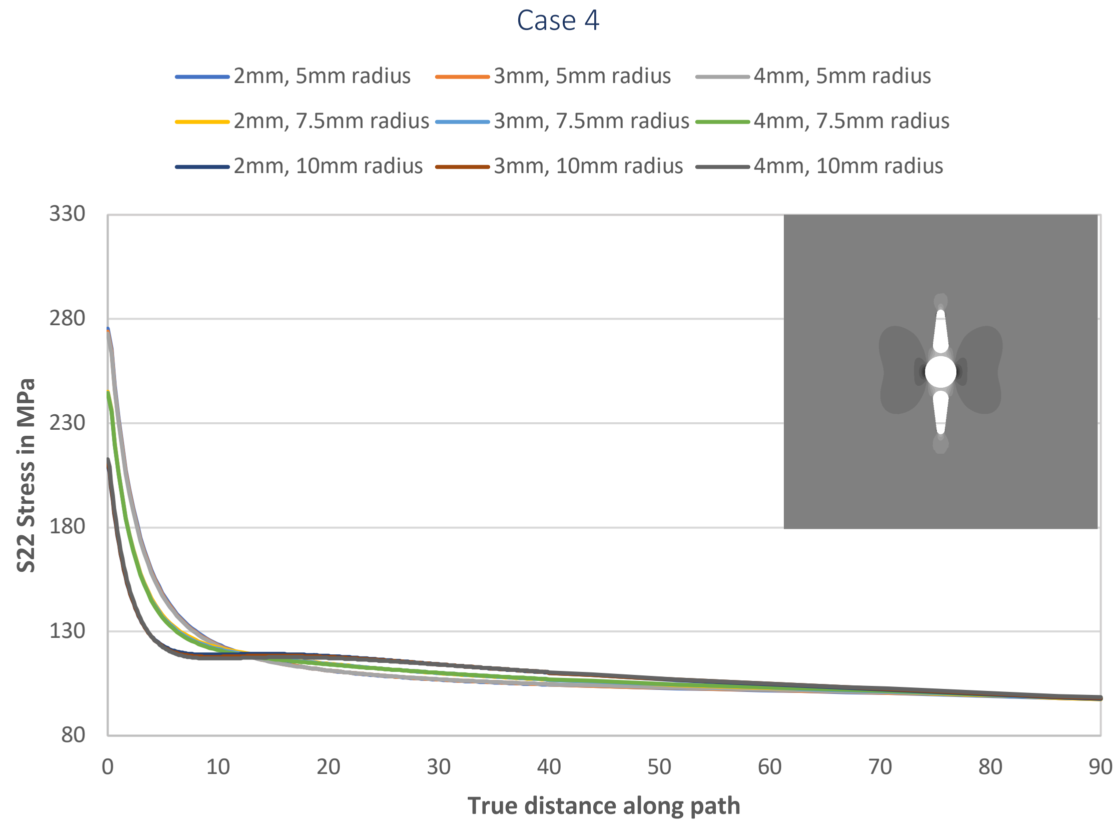


Figure 23: Stress along path for Case 4

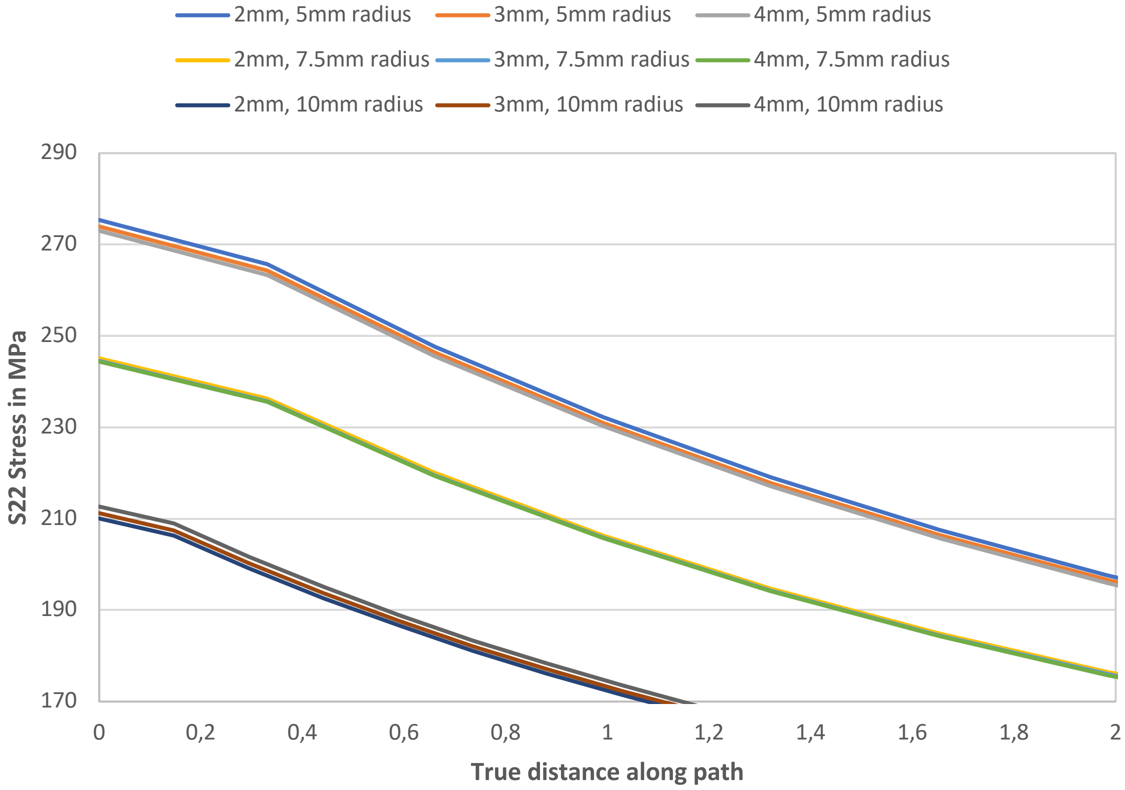


Figure 24*: Magnified view of stress along path for Case 4 for section 0 to 2*

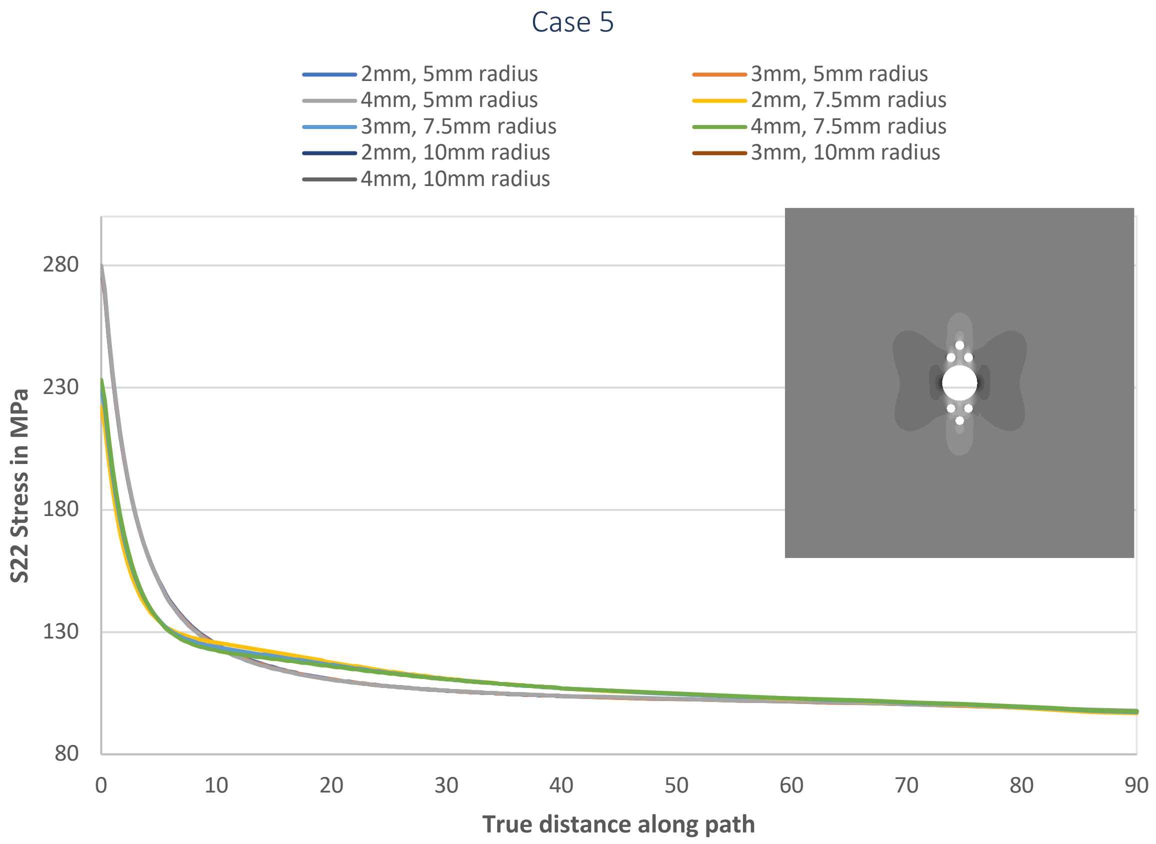


Figure 25: Stress along path for Case 5

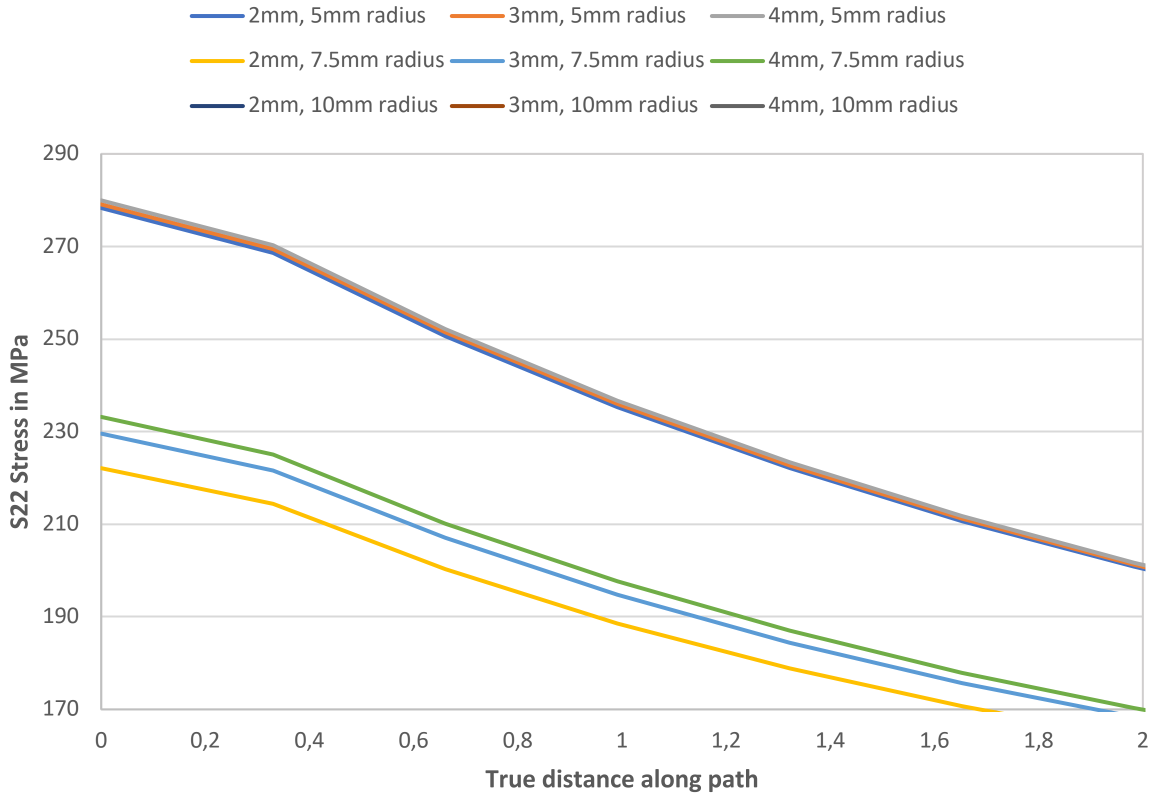


Figure 26: Magnified view of stress along path for Case 5 for section 0 to 2

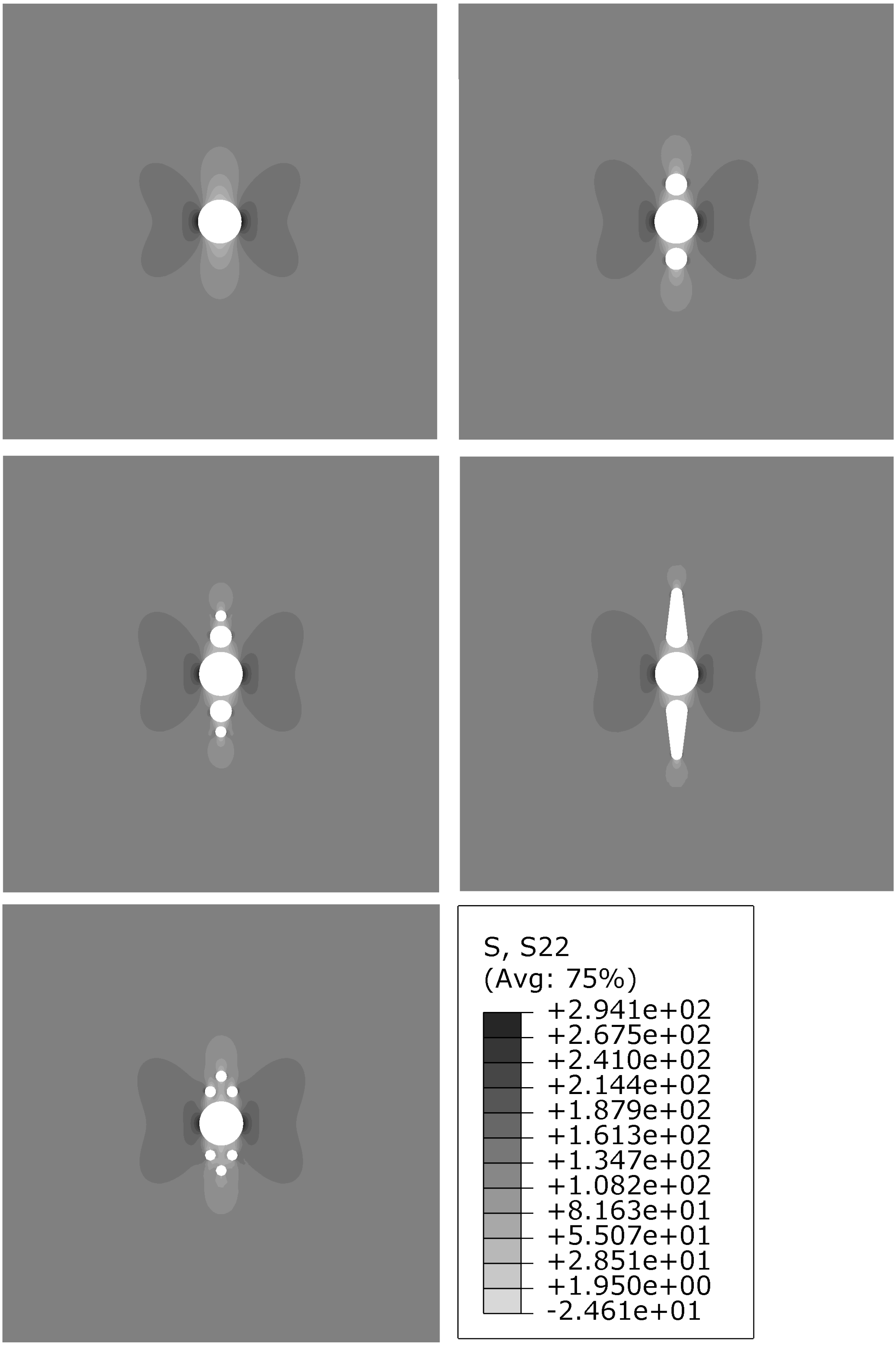


Figure 27: Comparison of Case 1-5

Fatigue analysis

After running the stress analysis, the following 5 geometries for fatigue testing were chosen. The reason for choosing only circular voids was to redistribute majority of the stress, without weakening the structure like the long void. The fillets of the specimens were modeled after ASTM standard. The smallest hole for case 4 and 5 is 70% of the larger holes on their respective geometry. Distance between each hole is kept constant at 2mm for all cases, as the dimensions shows.

Geometries

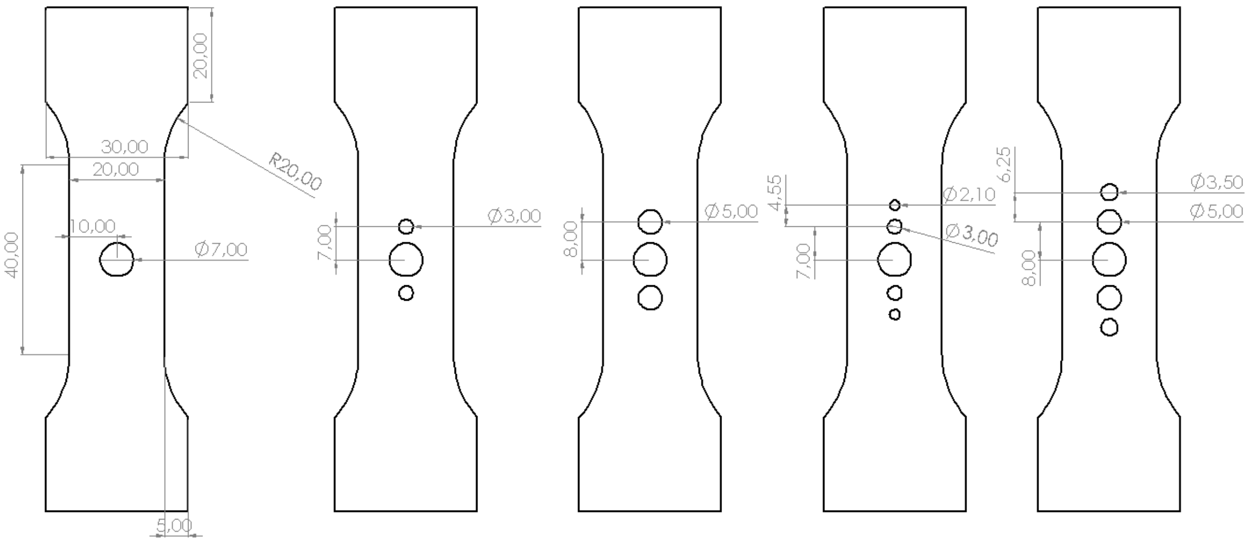
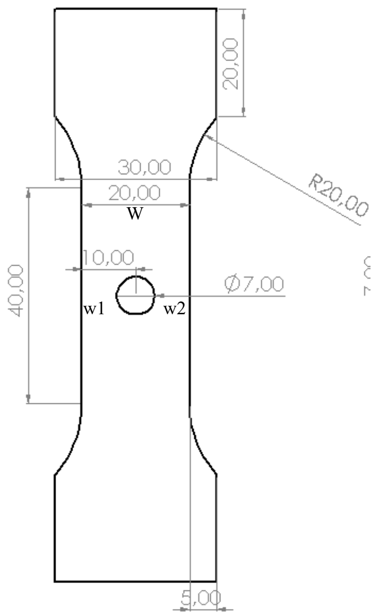


Figure 28: From left to right, Case 1, 2, 3, 4 and 5

Geometries for produced test specimens

The produced specimens for the fatigue test were so similar in size that the small variations were neglected, and the same load was applied for all the test.



W = width of the specimen's neck
at narrowest point. (20mm in the drawings)
w1 = width from hole and to the left edge. (6,5mm)
w1 = width from hole and to the right edge. (6,5mm)
t = thickness of the specimen (3mm)

Figure 29: Case 1 used to illustrate W, w1 and w2 measurements

C-1-1	
W	19,9mm
w1	6,5mm
w2	6,4mm
t	3,0mm

C-2-1	
W	19,9mm
w1	6,4mm
w2	6,5mm
t	3,0mm

C-3-1	
W	19,9mm
w1	6,4mm
w2	6,5mm
t	3,0mm

C-4-1	
W	20,0mm
w1	6,5mm
w2	6,6mm
t	3,1mm

C-5-1	
W	19,9mm
w1	6,4mm
w2	6,5mm
t	3,0mm

C-1-2	
W	19,9mm
w1	6,5mm
w2	6,4mm
t	3,0mm

C-2-2	
W	19,9mm
w1	6,5mm
w2	6,4mm
t	3,0mm

C-3-2	
W	19,9mm
w1	6,3mm
w2	6,4mm
t	3,0mm

C-4-2	
W	19,9mm
w1	6,5mm
w2	6,5mm
t	3,1mm

C-5-2	
W	19,9mm
w1	6,4mm
w2	6,5mm
t	3,0mm

C-2-3	
W	19,9mm
w1	6,5mm
w2	6,5mm
t	3,0mm

C-3-3	
W	19,9mm
w1	6,4mm
w2	6,5mm
t	3,0mm

C-5-3	
W	19,9mm
w1	6,4mm
w2	6,4mm
t	3,0mm

C-2-4	
W	19,9mm
w1	6,4mm
w2	6,4mm
t	3,0mm

C-4-4	
W	19,9mm
w1	6,3mm
w2	6,4mm
t	3,0mm

Tabell 3: Dimensions of produced test specimens

Test procedure FEA

The FEA for the 5 new geometries was conducted in the same manner as already described in "Detailed FEA procedures".

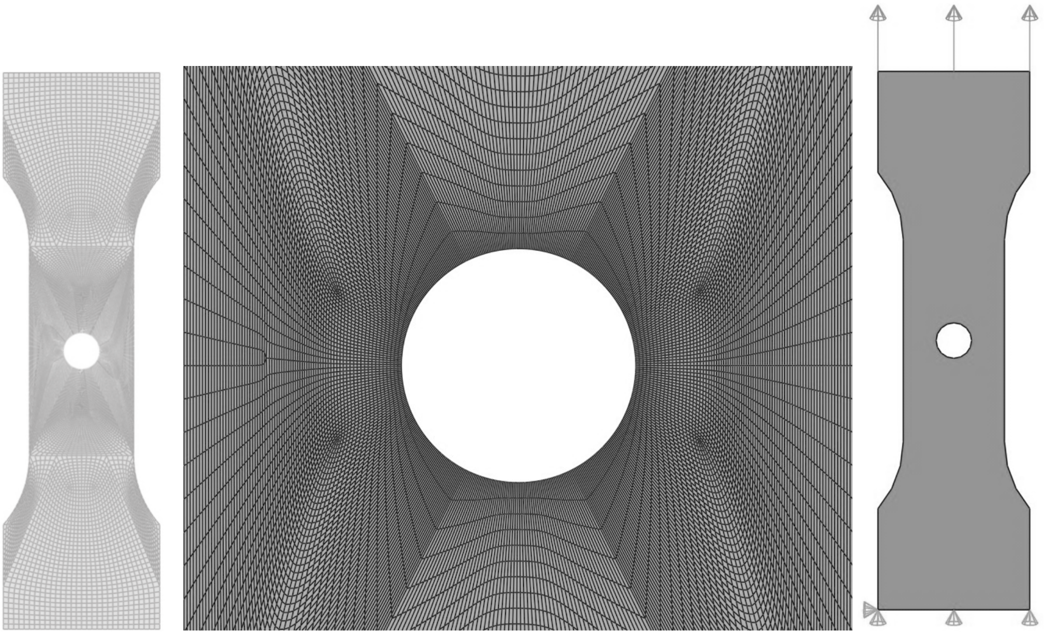


Figure 30: From left to right; Mesh, detailed view of mesh and BC & Loads for FEA

Procedure fatigue test

Before getting access to the fatigue lab at NTNU, safety courses were completed. First, a general safety course for the whole lab, then a specific course for the fatigue lab, then eventually a specific course for the Instron ElectroPuls E10000 fatigue machine with supervisor, Javad Razavi.

The fatigue test was conducted on 5754 aluminum alloy sheets. The specimens were water cut into the right geometry. The frequency for the fatigue machine was set to 40 Hz, the vertical shift was set to 3050 Newton and the amplitude was set to 3000 Newton. The added 50 newtons to the vertical shift made sure there was no slack in the machine when the amplitude got to negative 3000 newtons. All five geometries were tested with the same procedures and ran until failure. The test was stopped when the extension exceeded a certain limit, and is the reason why not all of the specimens are completely broken. This was neglected since the specimens were so deformed that it would only take a few repetitions more to reach complete separation. The repetitions until failure were obtained and are the basis for further discussion.

Figure 31: Computer setup while testing

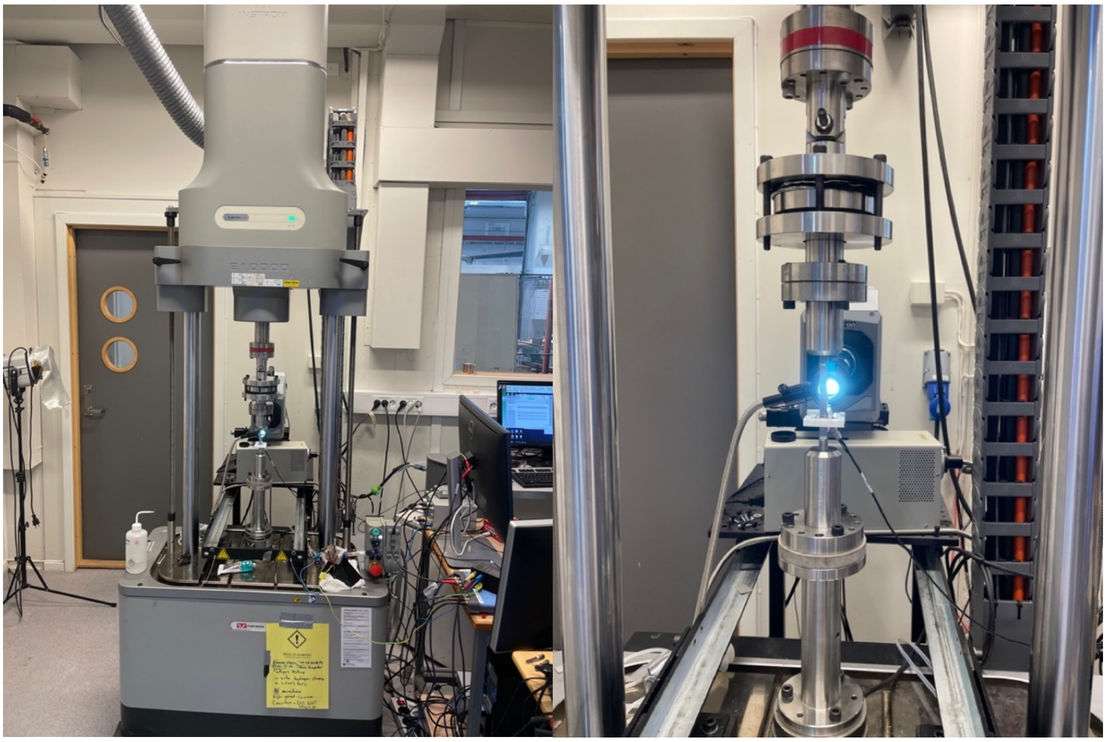


Figure 32: Instron ElectroPuls E10000

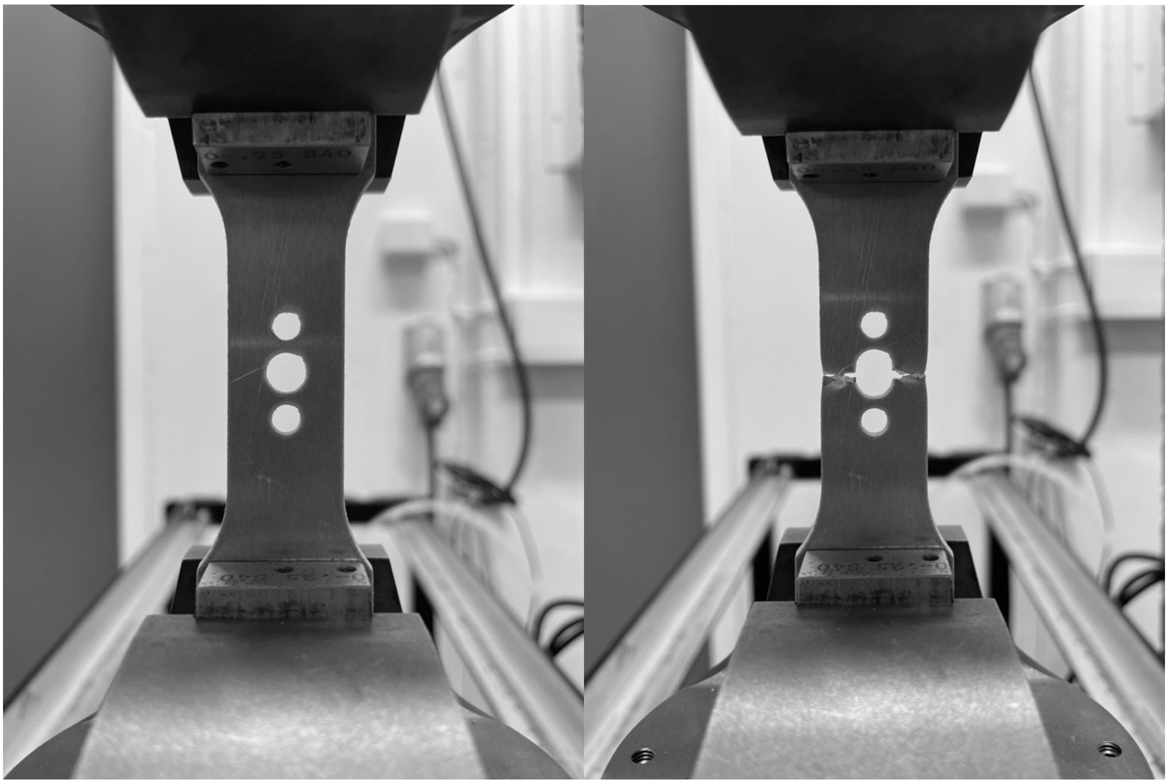


Figure 33: Case 3 before and after failure

FEA results

Maximum contour limit on the legend is set to the maximum occurring stress, which happens at Case 1 at 511.852 MPa for the S22 stress. Minimum limit is set to -14.9305 MPa, which is the lowest occurring stress, happening at Case 3.

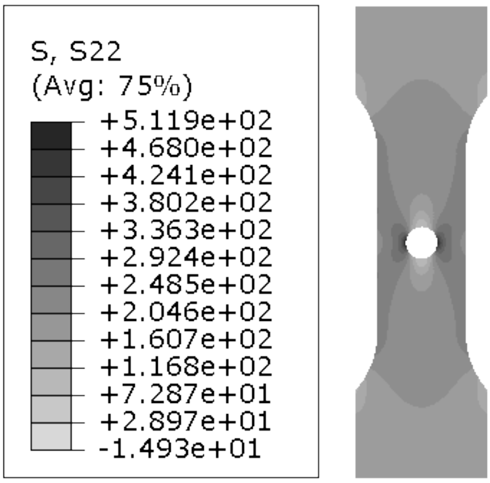


Figure 34: Case 1 and legend. Same legend limits used for all stress contours.

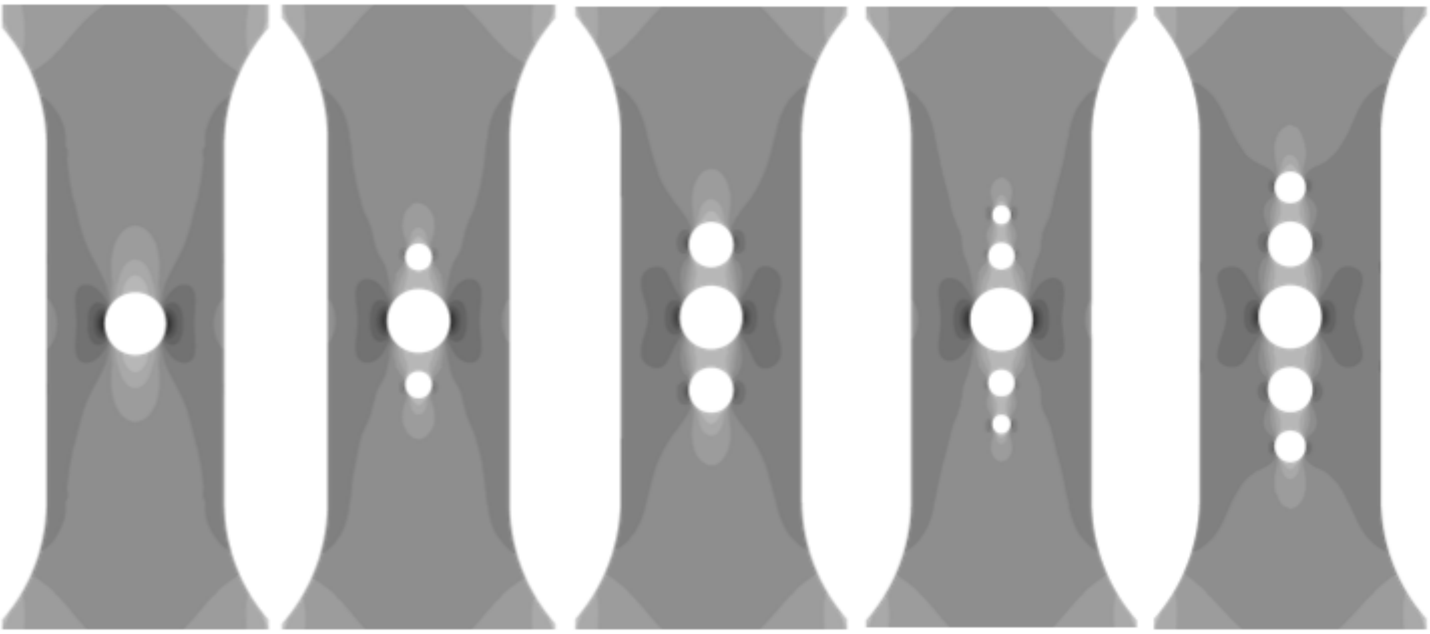


Figure 35: Stress contours of Case 1 to 5 from left to right

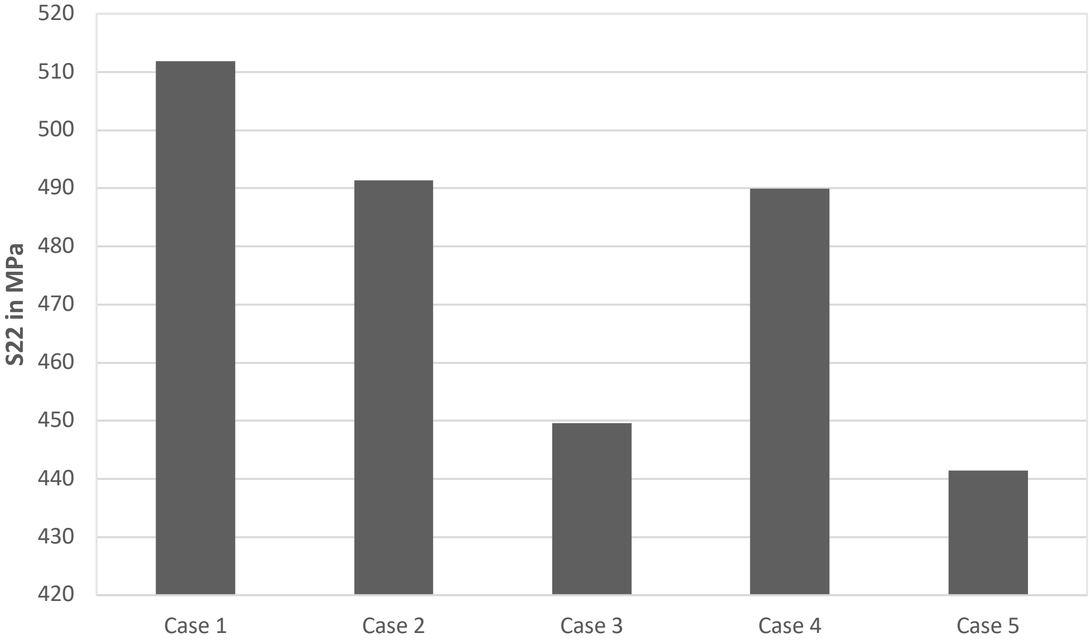


Figure 36: Max S22 stress from FEA for each case

Results fatigue test

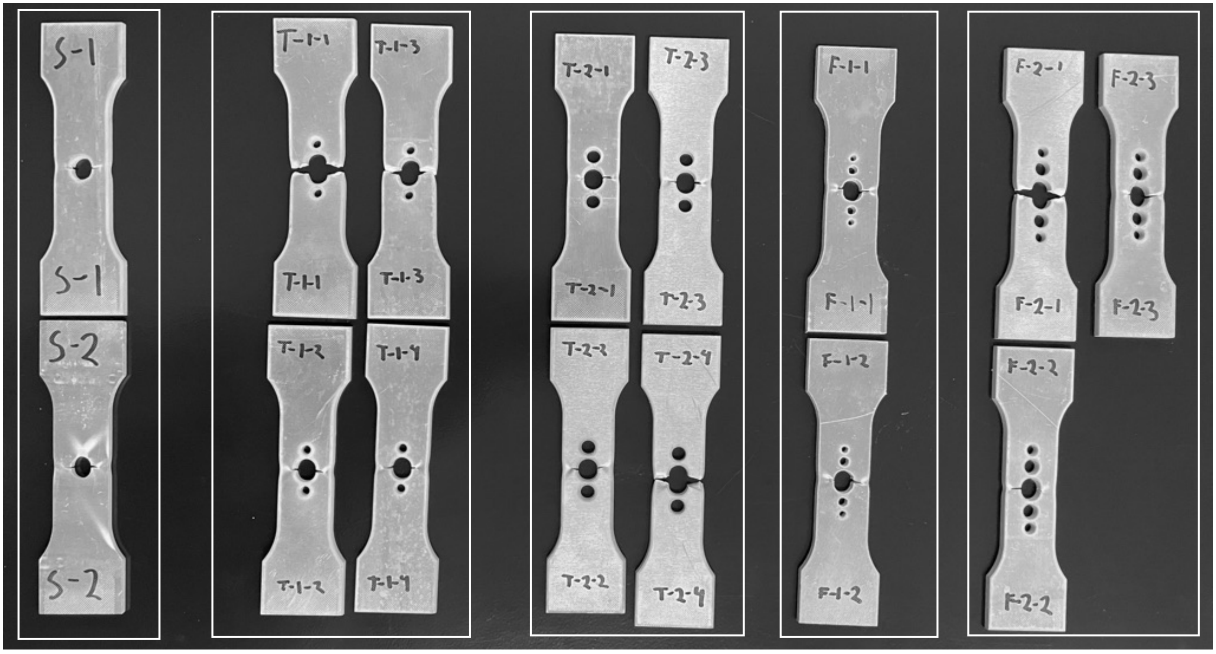


Figure 37: Case one to five from left to right. Each white box is one case.

Case 1		
C-1-1	42199	
C-1-2	40514	

Case 2		
C-2-1	47579	
C-2-2	36788	*
C-2-3	57140	*
C-2-4	42076	

Case 3		
C-3-1	80789	
C-3-2	76053	
C-3-3	67940	
C-4-4	47149	*

Case 4		
C-4-1	36525	**
C-4-2	32024	**

Case 5		
C-5-1	63782	*
C-5-2	86047	
C-5-3	60640	*

Table 4: Case name next to repetitions until failure

*Potential deviations

**Specimens had a coarse finish, and had a premature failure because the crack growth started earlier than it would on a fine finish.

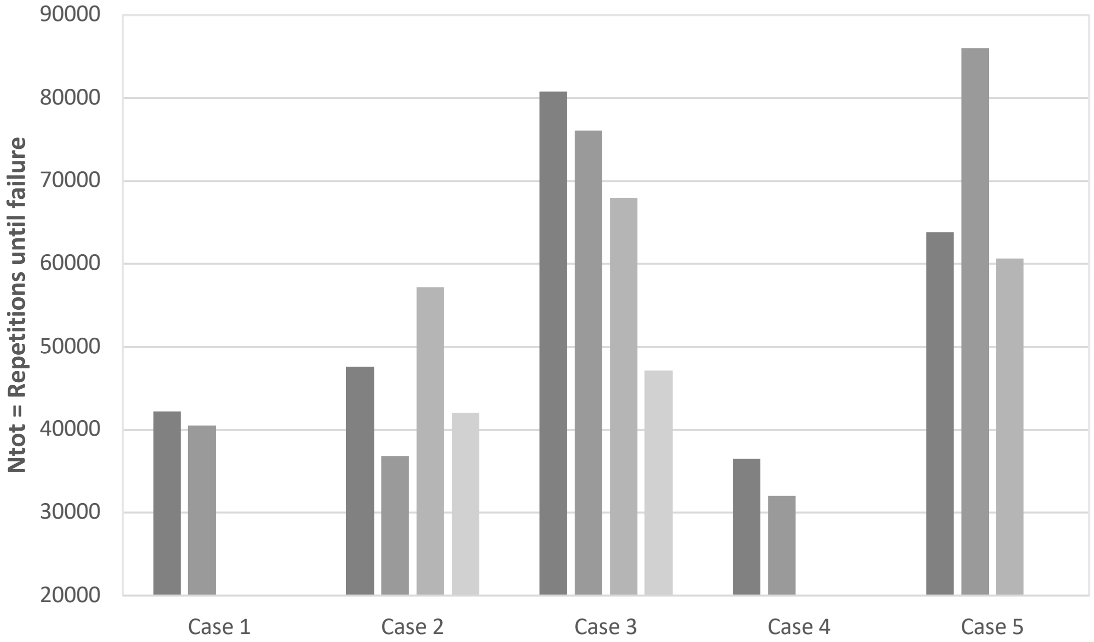


Figure 38: Bar plot showing repetitions until failure for each specimen for each case

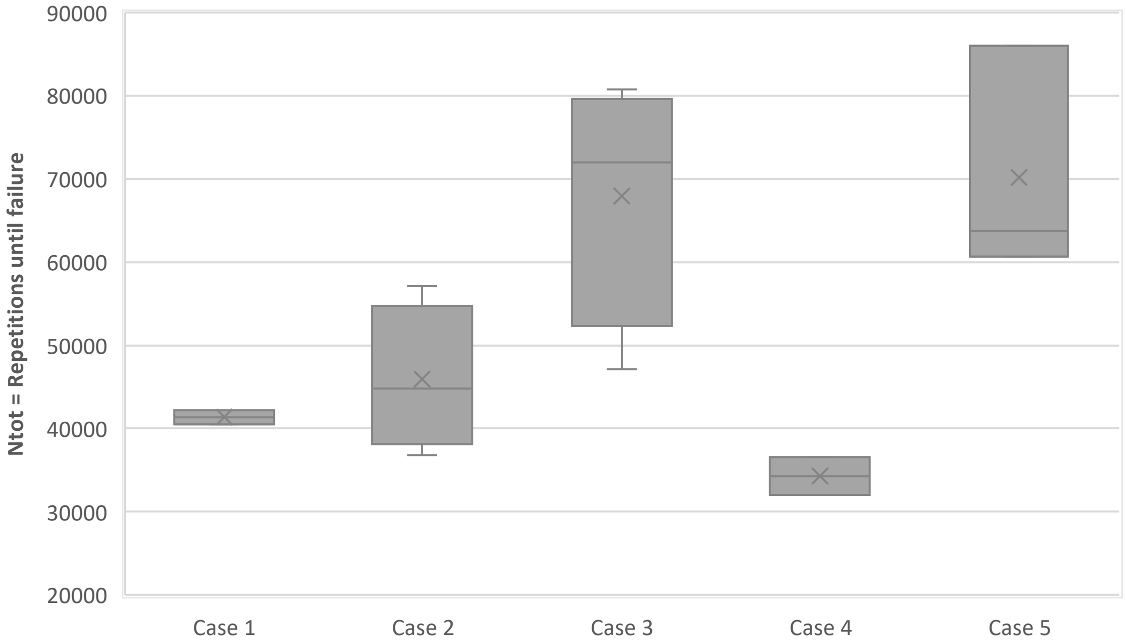


Figure 39: Whisker plot showing the minimum value, first quartile, median, third quartile and maximum value of Table 4

Second FEA Stress Analysis

For the second analysis in the spring of 2021, 9 geometries were created. Since several of the specimens in the first batch were damaged, a new batch was designed and produced. The geometries were decided alongside supervisor, Javad Razavi, and the aim was to further enhance our knowledge around the behavior of the tailored voids, as well as remove the flawed surface finish. We decided to ditch the extra hole in the 5 hole configuration for simplicity. The two extra holes on top and below did not yield much extra effect for the added complexity. We also exaggerated the distance variation between the holes to further provoke a larger response in the stress redistribution.

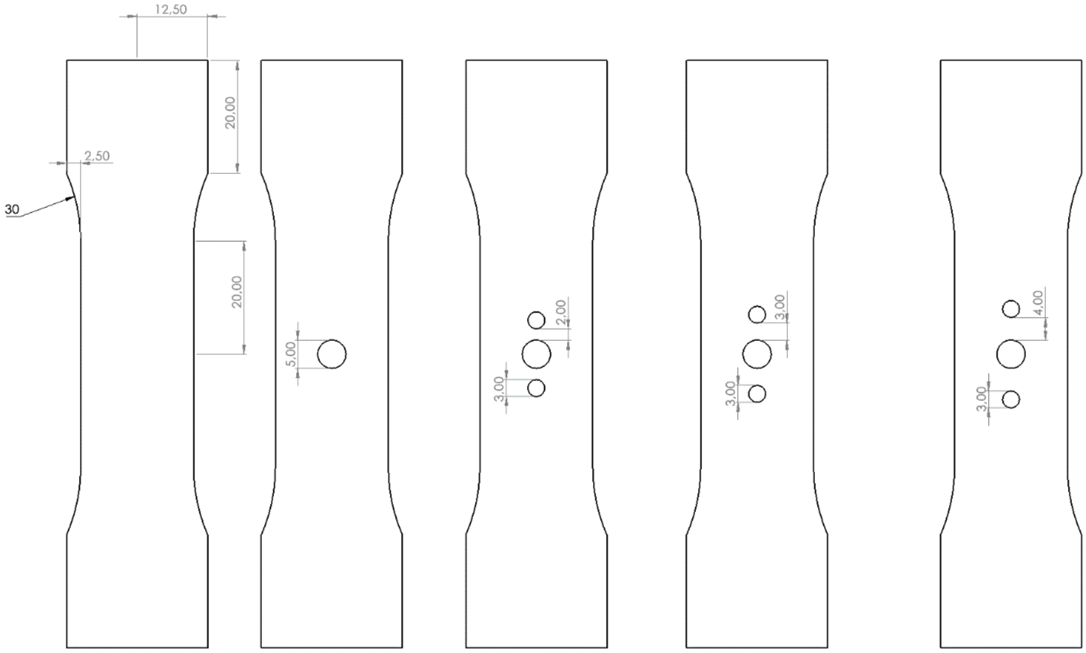


Figure 40: From left to right; Plain, Notch-1, Notch-3-1, Notch-3-2, Notch-3-3

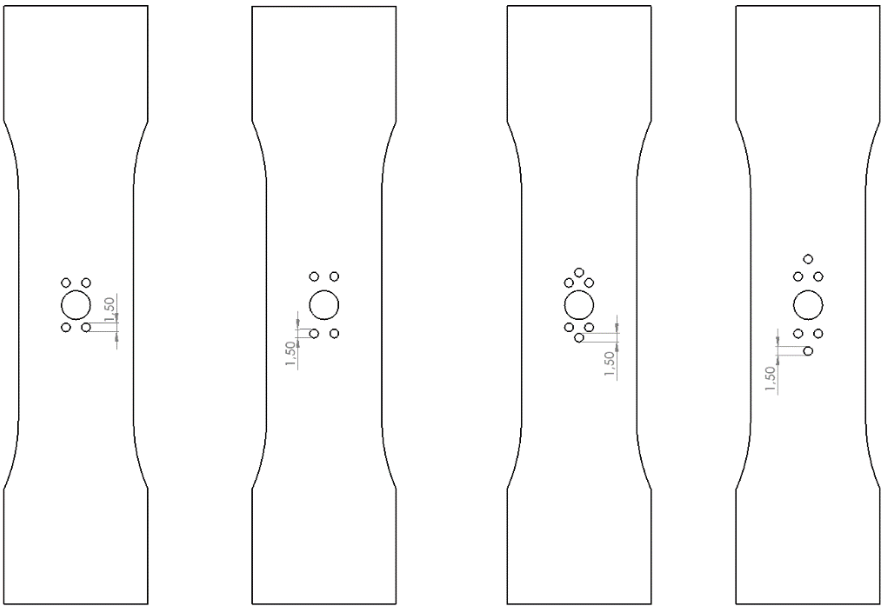


Figure 41: From left to right; Notch-5-1, Notch-5-2, Notch-7-1, Notch-7-2

The FEA for the new geometries was conducted in the same manner as already described in "Detailed FEA procedures".

Results for second FEA Stress Analysis

Results are discussed in the discussion chapter, but should be studied carefully by reader.

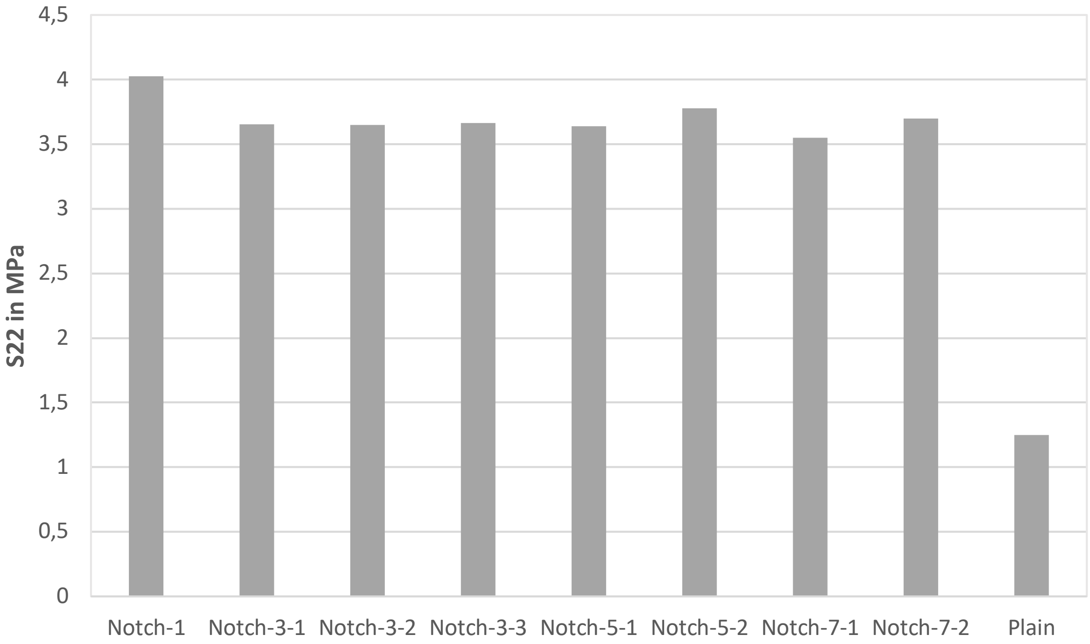


Figure 42: Bar plot showing max S22 stress in MPa at center hole for each case

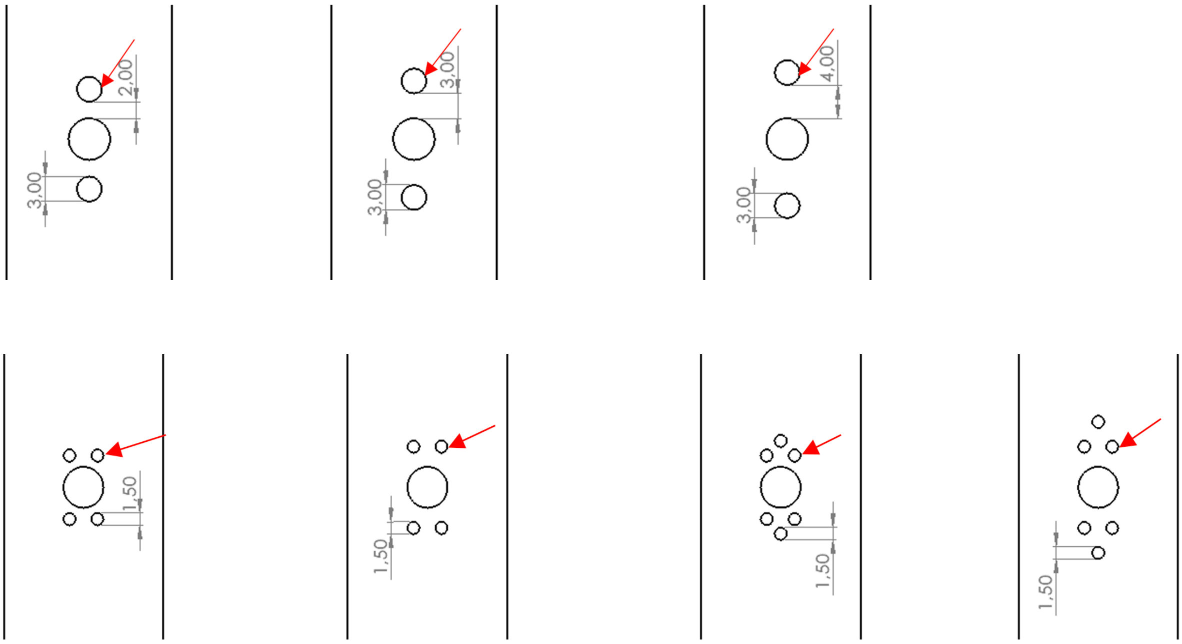


Figure 43: Red arrows showing the location for the highest stresses in the added tailored voids

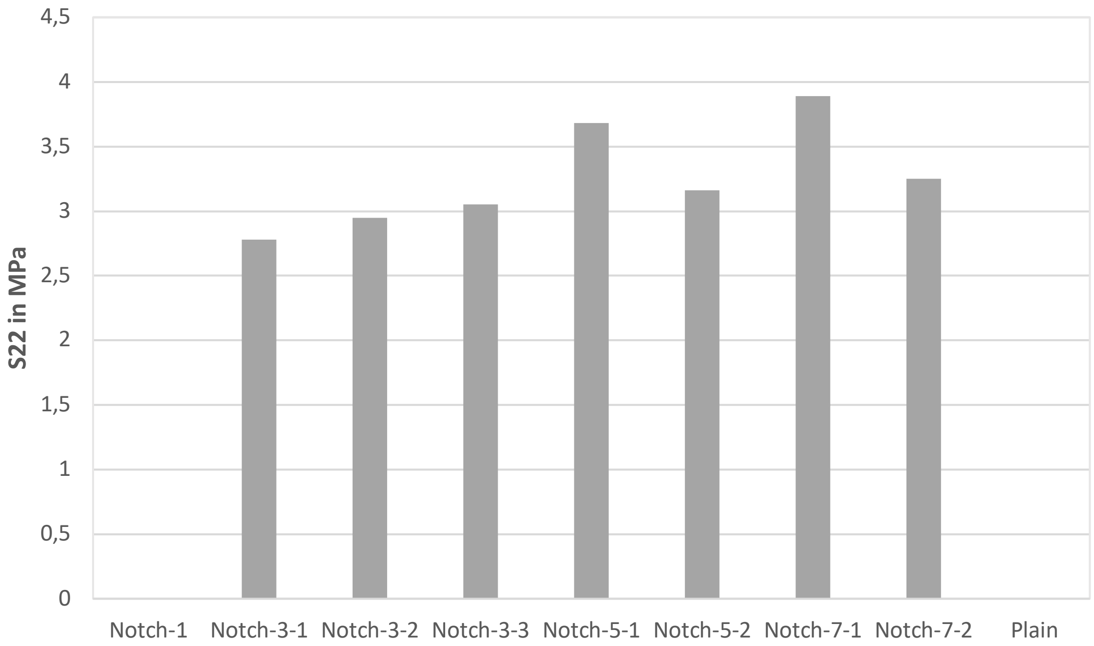


Figure 44: Bar plot showing max S22 stress in MPa at the most critical added notch

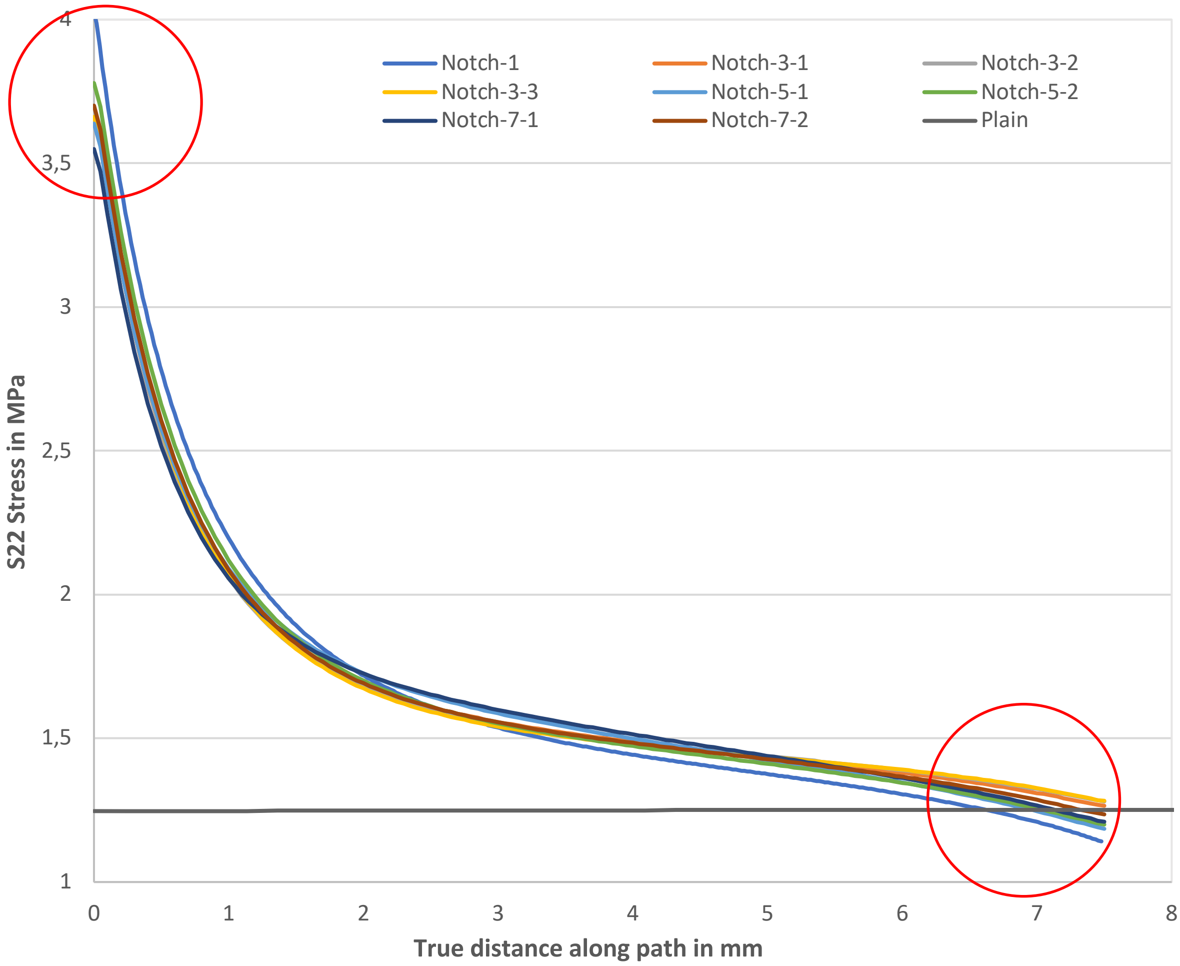


Figure 45: Stress along path in mm. Red circles show areas that will be zoomed in further

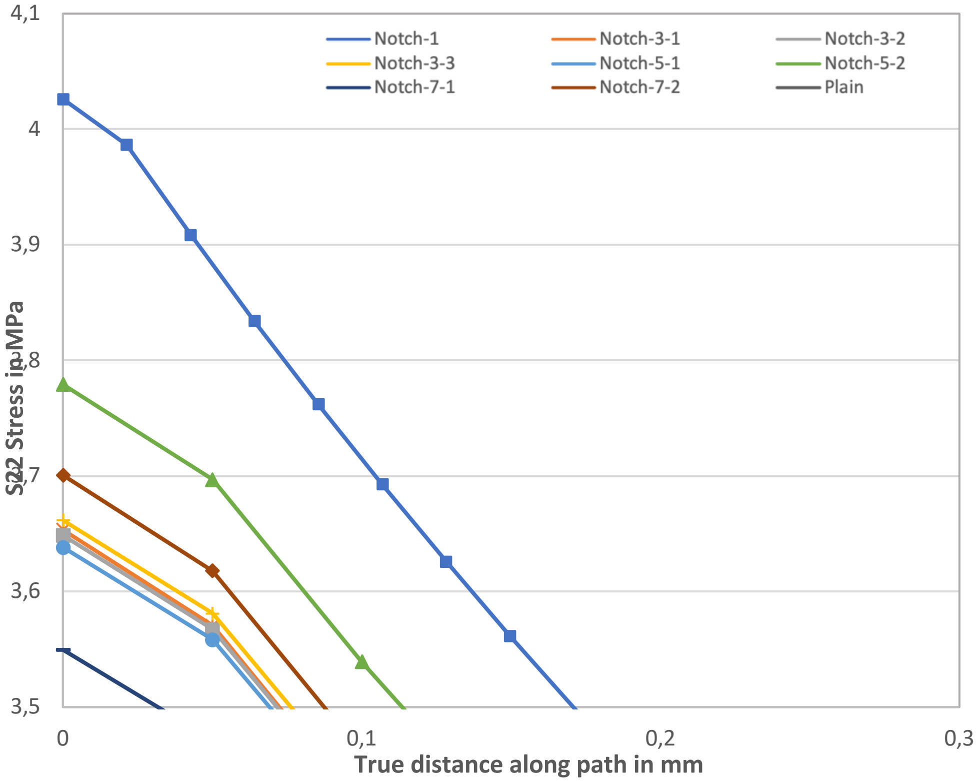


Figure 46: Magnified view of stress along path for section 0-3 located closest to the notch

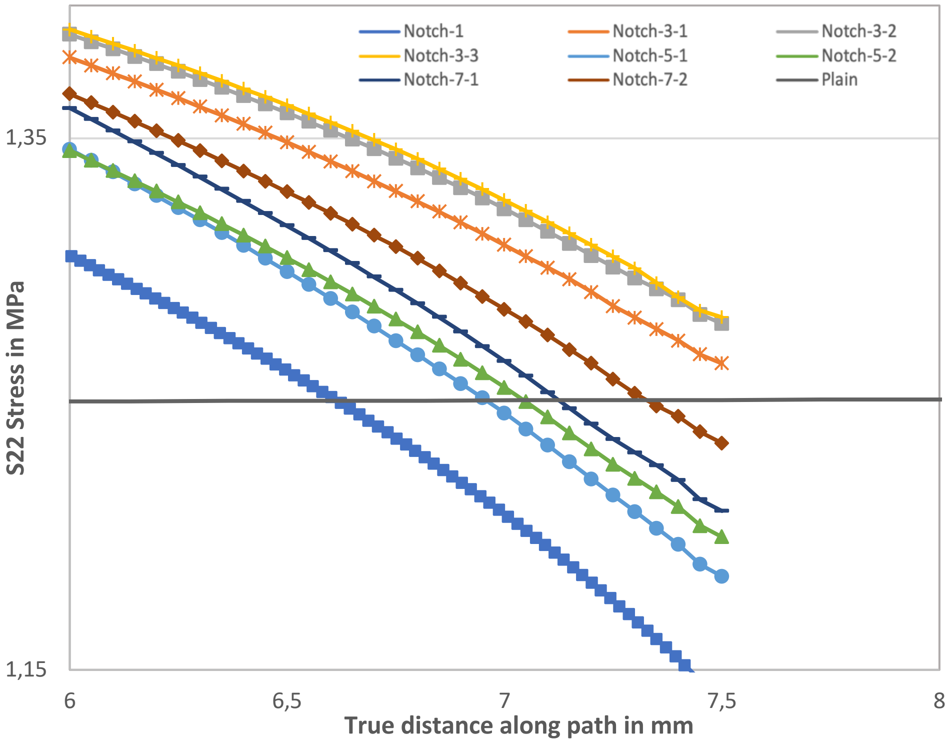


Figure 47: Magnified view of stress along path for section 6-8 located furthest away from the notch

Maximum contour limit on the legend is set to the maximum occurring stress, which happens at Notch-1 at 4.041 for the S22 stress. Minimum limit is set to 0.029975, which is the lowest occurring stress, happening at Notch-3-3.

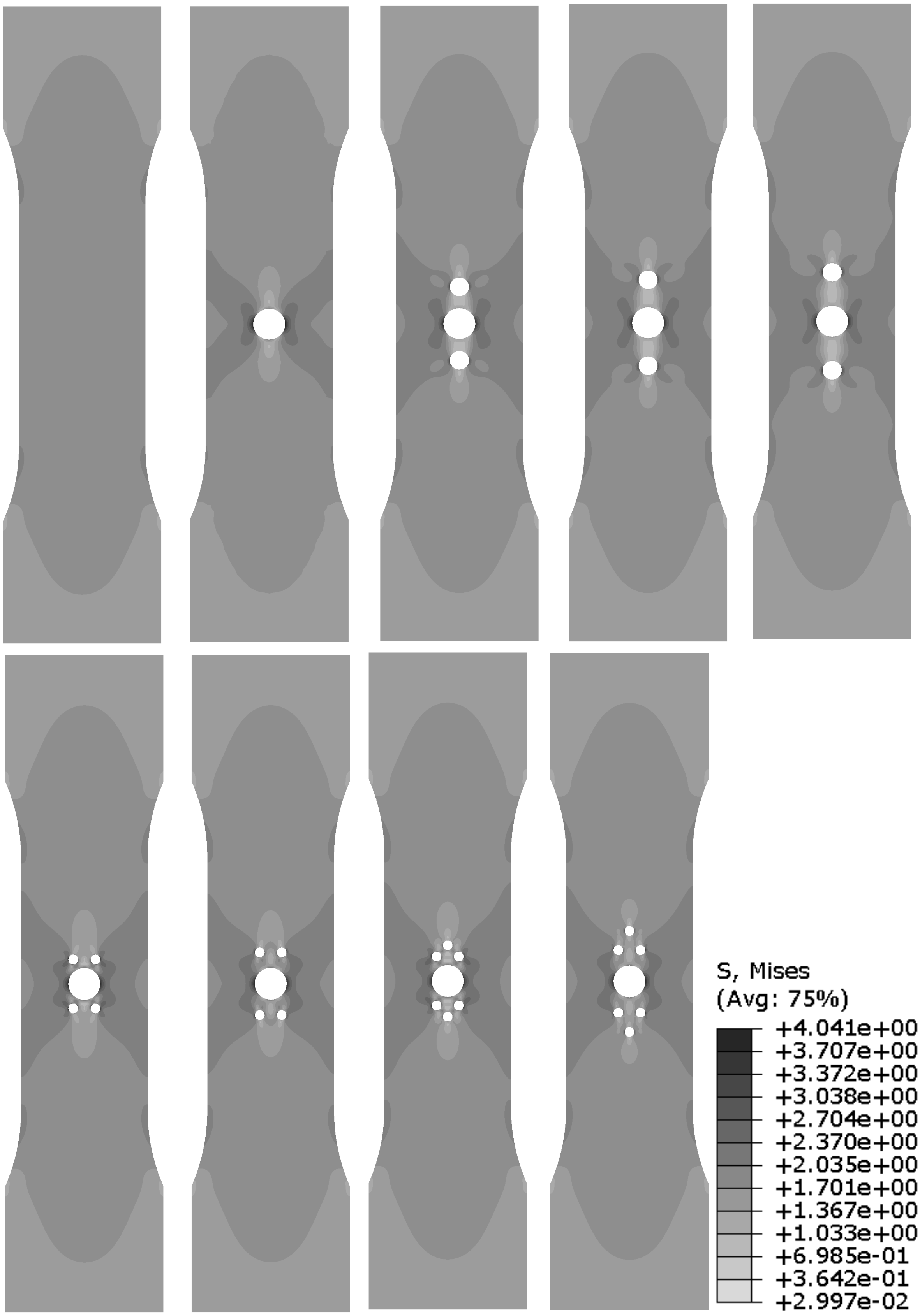


Figure 48: Stress contours

Second Fatigue analysis

For the second fatigue test, a new hydraulic machine was used, unlike the electric driven fatigue machine in the first fatigue test. A new introduction course for MTS-Series 809 Axial/torsional test system – 100KN fatigue machine was completed with supervisor, Javad Razavi.

Maximum applied load: 6 kN
Loading ratio, R: 0.01
Loading frequency: 20 Hz
Number of repetitions: two specimens for each geometry

Material: 5754 Aluminum alloy
Fabrication technique: CNC cutting
Tensile properties: E=70GPa, ν=0.35, UTS=300MPa

The fatigue test was conducted on sheets of aluminum. The specimens were water cut into the right geometry, and the holes were precisely drilled. All geometries were tested with the same procedures and ran until failure. The test was stopped when the extension exceeded a certain limit. The repetitions until failure were obtained and are the basis for further discussion.

Figure 49: Close up picture of specimen prepared for testing

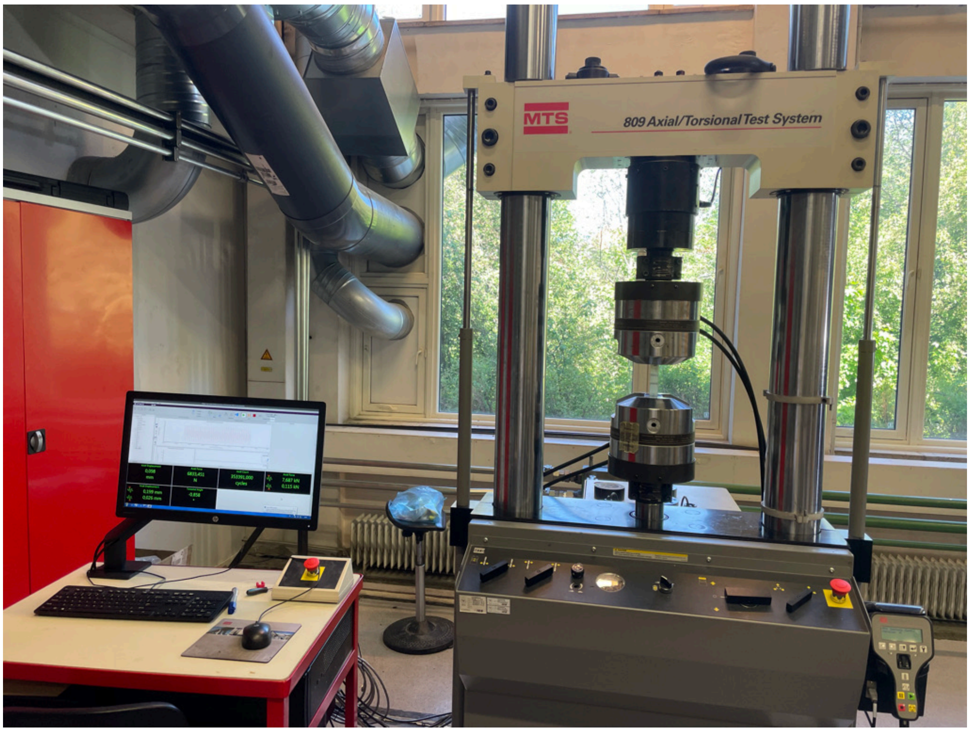


Figure 50: Fatigue machine MTS-Series 809 Axial/torsional test system – 100KN

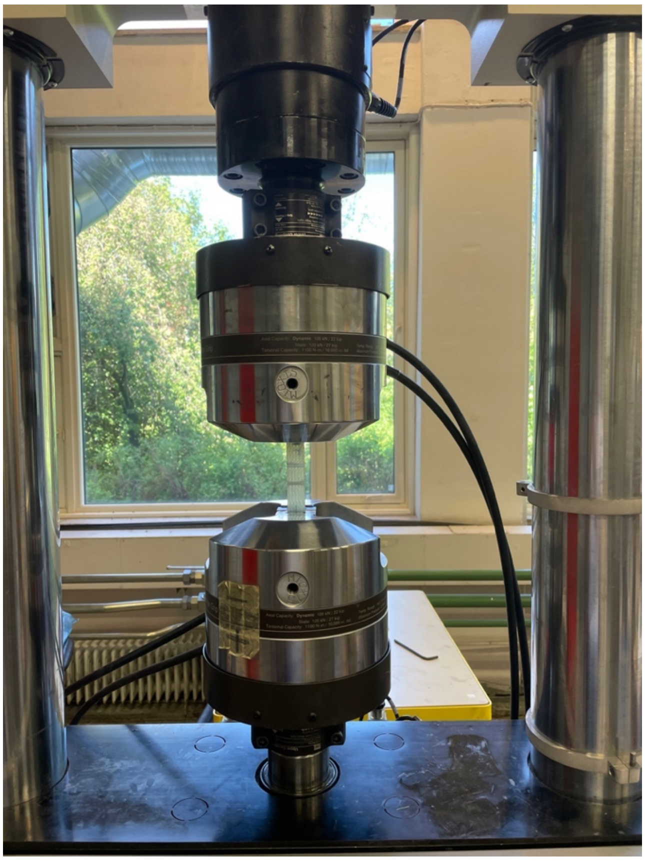


Figure 51: Magnified view of specimen fastened in both grippers

Figure 52: Control panel controlling gripping force. Has to be tuned to not be damaged while gripping

Results for Second Fatigue analysis

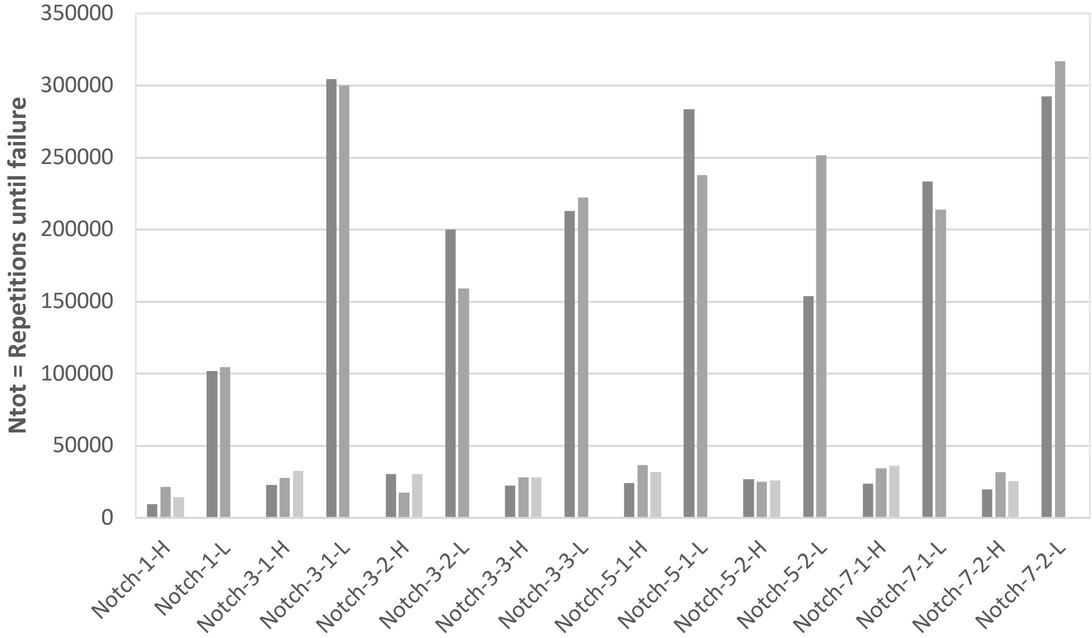


Figure 53: Bar plot showing repetitions until failure for each specimen

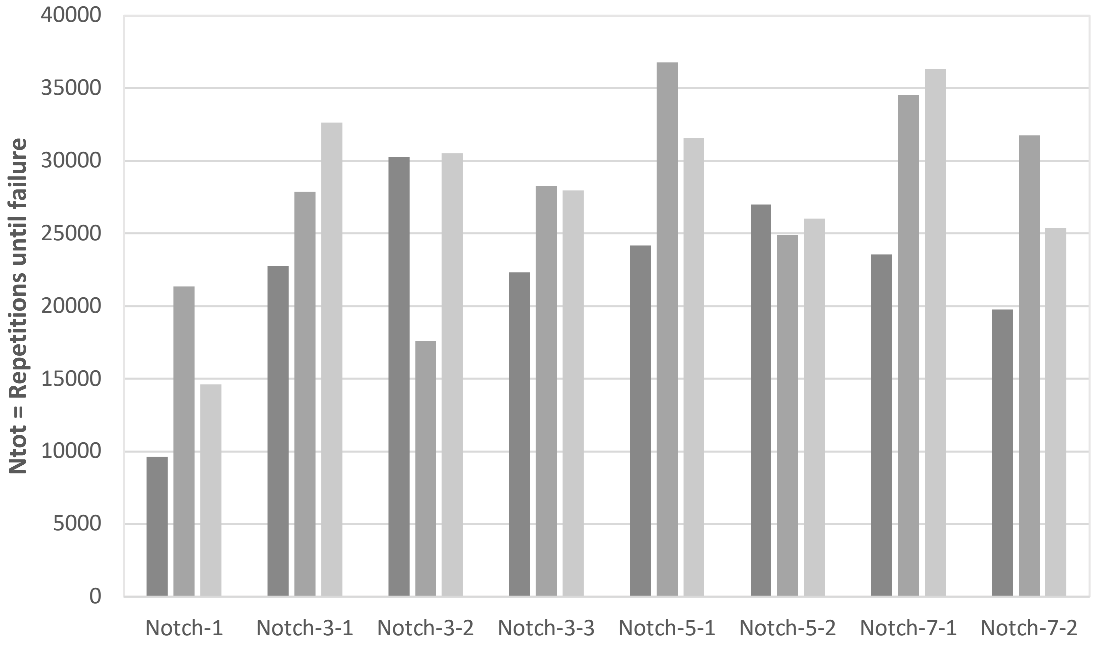


Figure 54: Bar plot showing repetitions until failure for only 200 MPaa

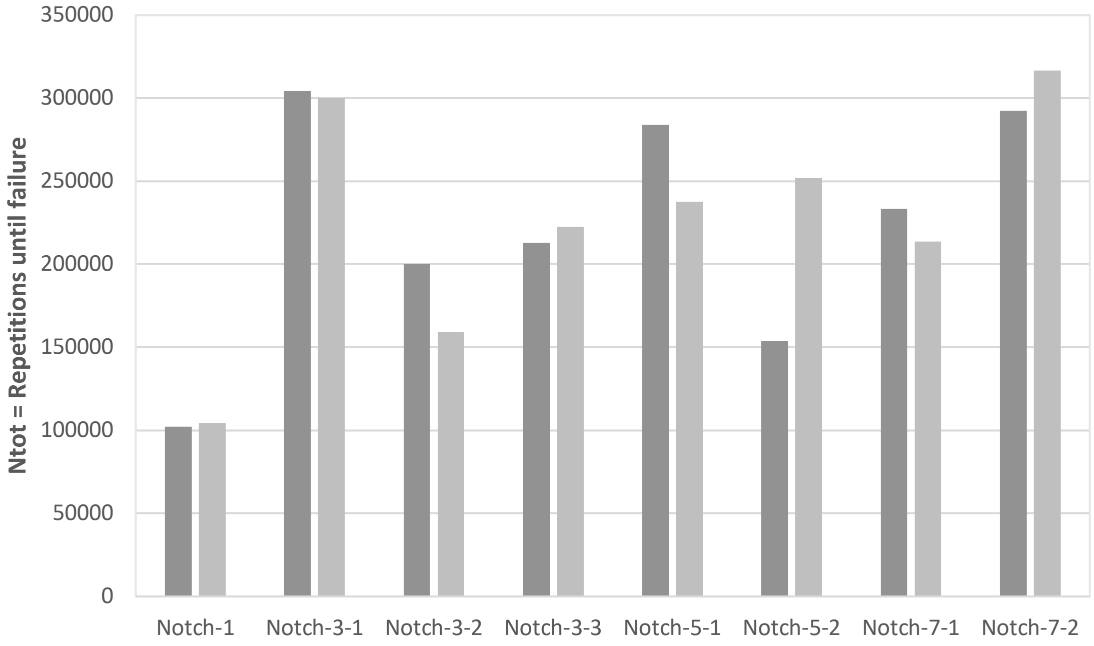


Figure 55: Bar plot showing repetitions until failure for only 150 MPa

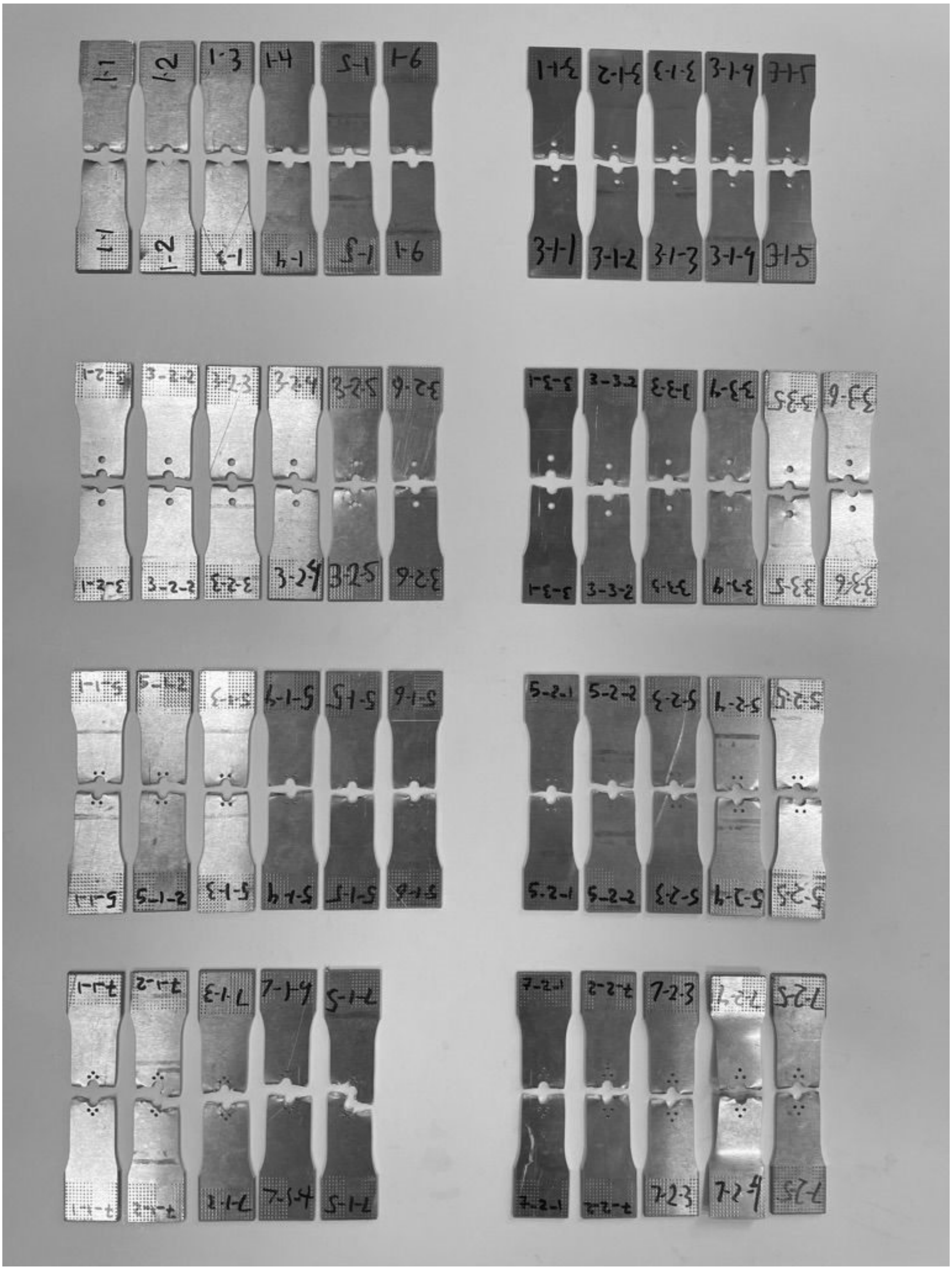


Figure 56: Showing all test specimens after failure

Lattice structure

For the final part of the thesis, we utilize all the knowledge acquired as inspiration when implementing lattice structures in the design. When the tailored voids increased in size, the critical maximum S22 stress at the notch tip decreased, up until the stress in the added voids was higher than the center hole. However, it would be safe to assume that the resistance against fatigue for multi axial loading would take a hit when the material is removed. We want to redistribute the stress as much as possible, while still performing well under multi axial loading and under single axial loading with compression. Therefore, a lattice structure geometry was created inside two elliptical half moons. The goal was to get as much stress redistribution as possible, without the negative side effects.

To create lattice structures, the software nTopology was utilized. An educational license was retrieved and the software was learned. The support of PhD Candidate, Zhuo Xu “Loker” was crucial for the creation of the lattice structures.

The steps taken during the creation of the lattice structures will be documented under the section called “Detailed nTopology procedures”.

Figure 57: 3d view of dogbone-specimen after lattice structure was added in nTopology

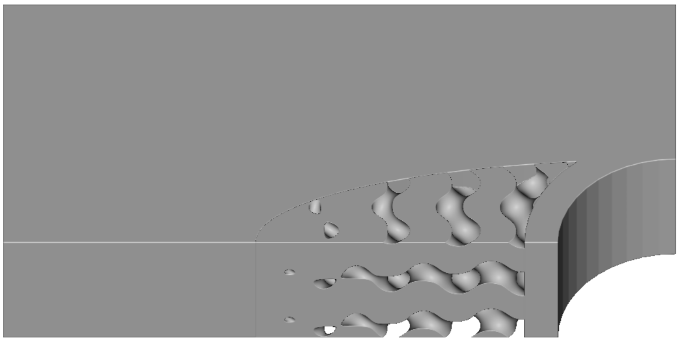


Figure 58: Zoomed cut through view of 1/4 section. 1/4 was used to minimize computer requirements

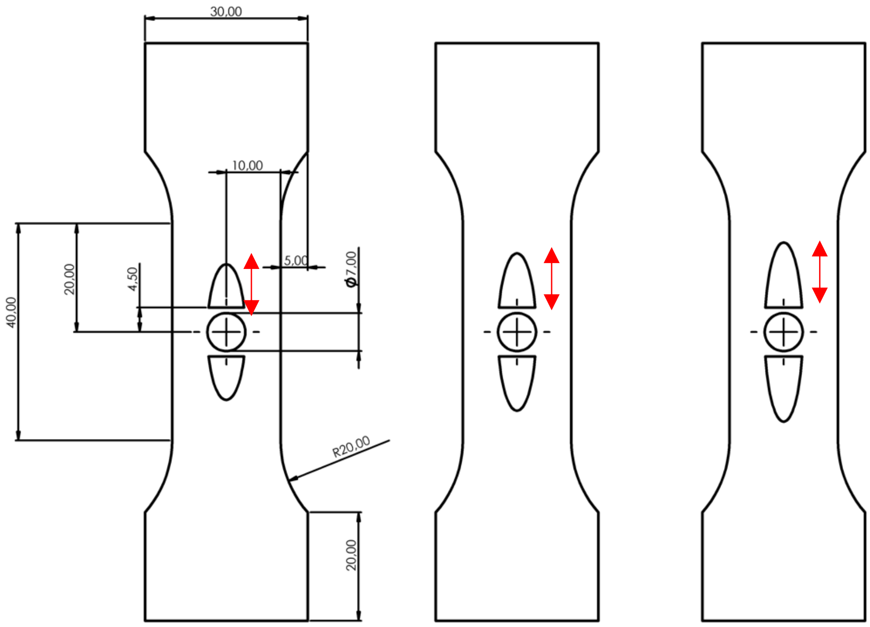


Figure 59: Red arrow shows design variable for the three geometries

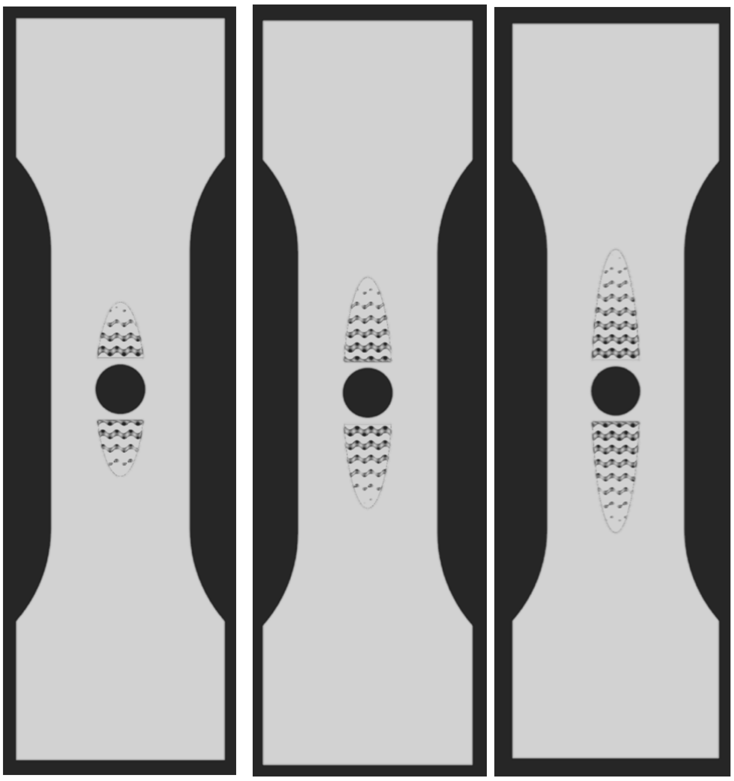


Figure 60: Specimens after lattice structure was added with nTopology

Distance between circle and half ellipse: 1mm (constant)
Length of half ellipse: 8mm, 12mm, 16mm

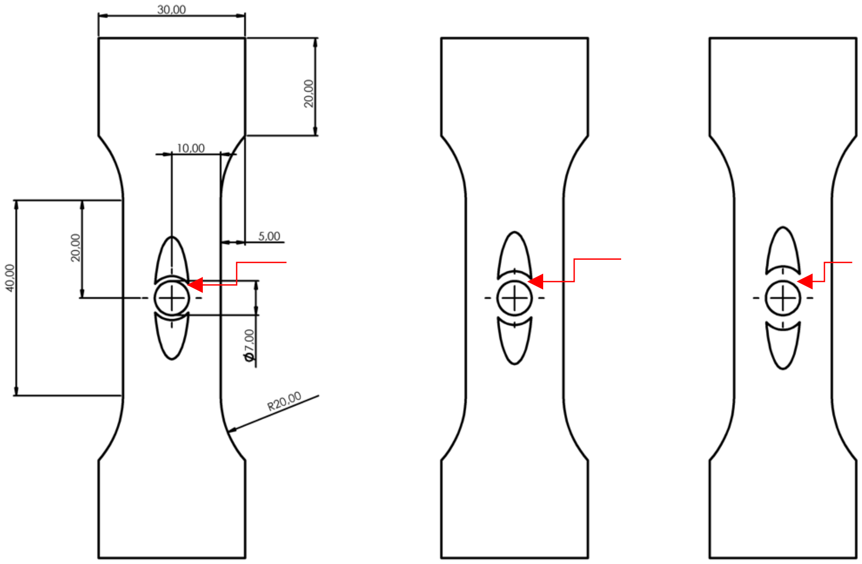


Figure 61: Red arrow showing design variable

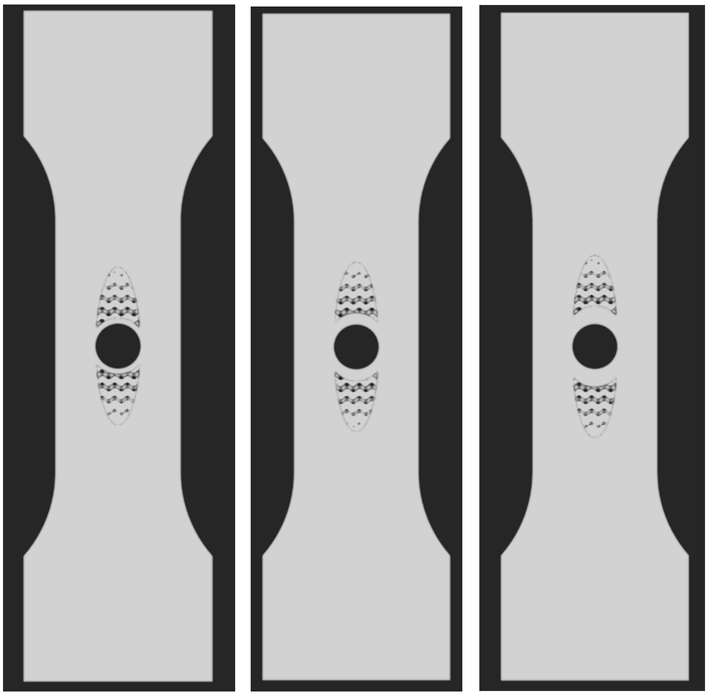


Figure 62: Specimens after lattice structure was added with nTopology

Distance between circle and half ellipse: 1mm, 2mm, 3mm
Length of half ellipse: 8mm (constant)

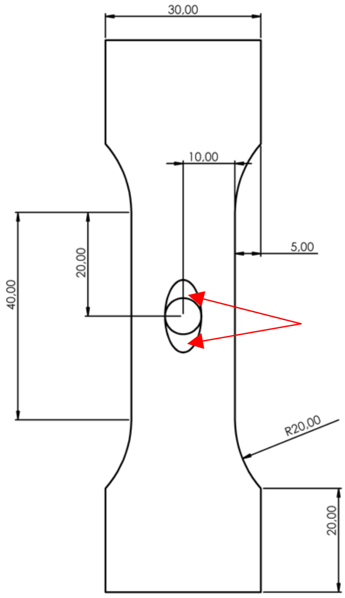


Figure 63: Red arrow showing voids with various porosity

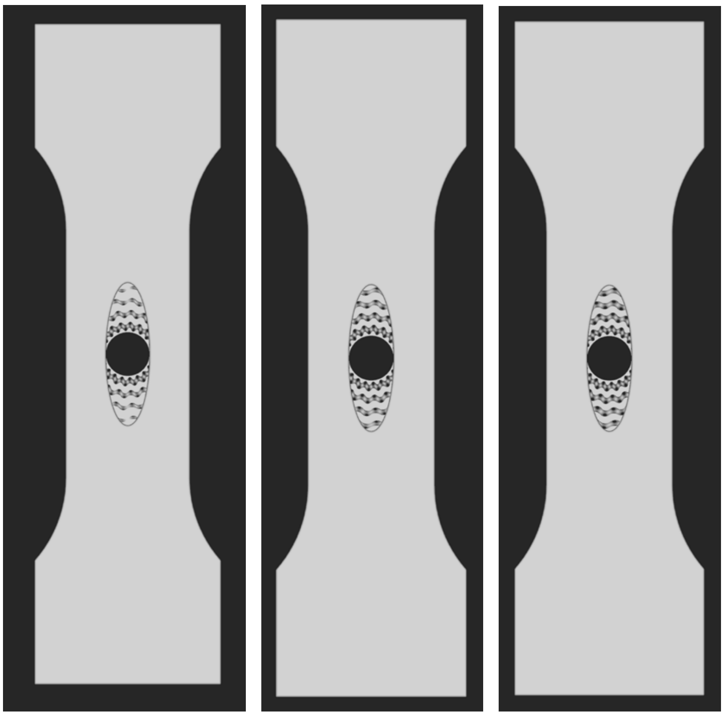


Figure 64: Specimens after lattice structure was added with nTopology

Porosity
Design 1: from 50% to 0%
Design 2: from 50% to 15%
Design 3: from 50% to 30%

Detailed nTopology procedures

The production of the gyroid was performed with nTopology 3.1.2 software. The steps taken throughout are provided to be able to replicate the study in the future; in addition, pictures are provided to show the building blocks, which is the workflow for nTopology.

1. Start nTopology:
Start – nTopology 3.1.2

2. 2. Create file name:
   File: Save as, [Project], Save

3. Add part block:
Module: Add description
   Name: «Whole_part»
   Add block: Import part
   Path: Select fitting path
   Heal CAD: Basic
   Check Model: Check

Repeat for all parts that will be imported

4. Add Implicit Body Block
   Name: «Implicit Body_1»
   CAD Body: Select the proper part
   Tolerance: 0.01 mm

Repeat for all parts that will be converted to implicit body

5. Add TPMS Block
   Name: «Implicit Body_2»
   Body: Select the proper part
   Cell size: 2x2x2 mm
   Approx. bias length: Select ramp function (will be added in next block)

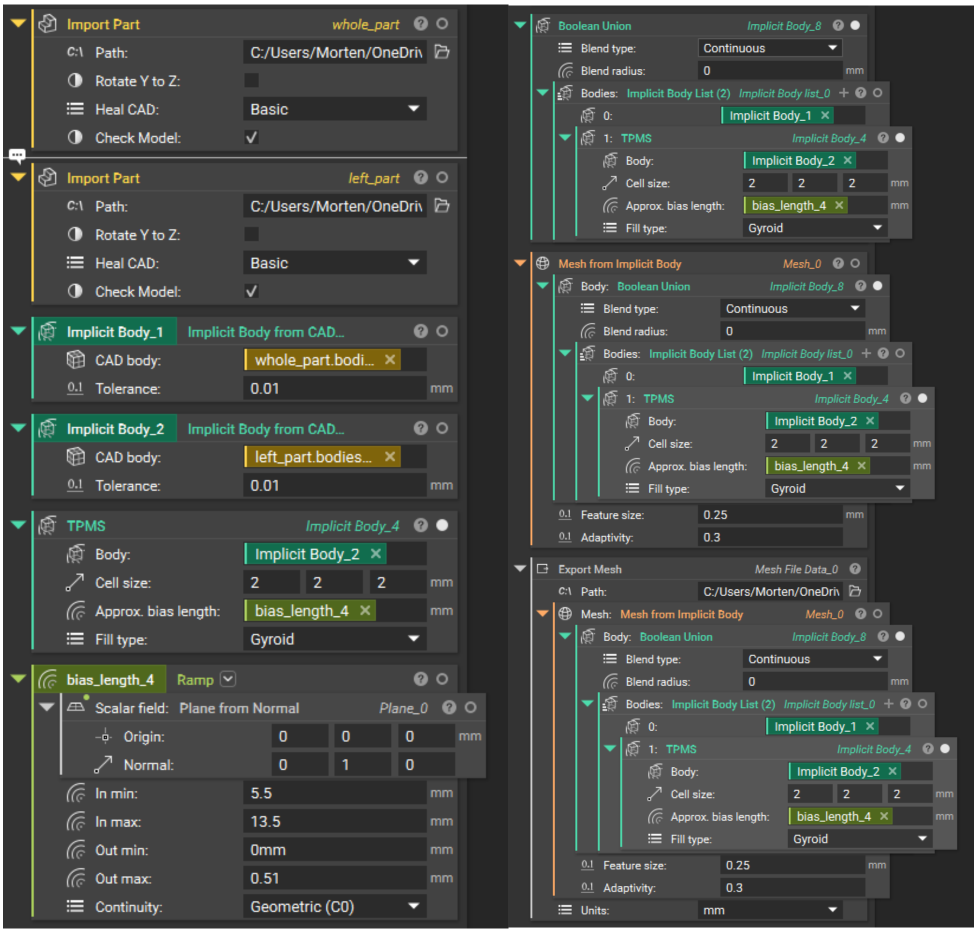
6. Add Ramp function
   Name: «Implicit_lenght_3»
   Scalar field: Plane from normal
      Origin: 0,0,0. Normal: Make a normal function that corresponds with the correct direction
   In min: 4.5 mm
   In max: 12.5 mm
   Out Min: 0mm
   Out Max: 0.51
   Continuity: Geometric

7. Add Boolean Union
   Blend type: Continuous
   Blend radius: 0 mm
   Body 0: Implicit Body_1
   Body 1: Add TPMS

8. Mesh from Implicit Body
   Body: Select the Boolean Union
   Feature size: 0.25
   Adaptivity: 0.3

9. Export Mesh
   Path: Choose suited path
   Units: mm

Done. Mesh is exported and will be further developed to generate a newly optimized part.



Import Part — whole_part
Path: C:/Users/Morten/OneDriv
Rotate Y to Z: ☐
Heal CAD: Basic
Check Model: ✓

Import Part — left_part
Path: C:/Users/Morten/OneDriv
Rotate Y to Z: ☐
Heal CAD: Basic
Check Model: ✓

Implicit Body_1 — Implicit Body from CAD...
CAD body: whole_part.bodi...
Tolerance: 0.01 mm

Implicit Body_2 — Implicit Body from CAD...
CAD body: left_part.bodies...
Tolerance: 0.01 mm

TPMS — Implicit Body_4
Body: Implicit Body_2
Cell size: 2 2 2 mm
Approx. bias length: bias_length_4 mm
Fill type: Gyroid

bias_length_4 — Ramp
Scalar field: Plane from Normal — Plane_0
Origin: 0 0 0 mm
Normal: 0 1 0
In min: 5.5 mm
In max: 13.5 mm
Out min: 0mm mm
Out max: 0.51 mm
Continuity: Geometric (C0)

Boolean Union — Implicit Body_8
Blend type: Continuous
Blend radius: 0 mm
Bodies: Implicit Body List (2) — Implicit Body list_0
0: Implicit Body_1
1: TPMS — Implicit Body_4
Body: Implicit Body_2
Cell size: 2 2 2 mm
Approx. bias length: bias_length_4 mm
Fill type: Gyroid

Mesh from Implicit Body — Mesh_0
Body: Boolean Union — Implicit Body_8
Blend type: Continuous
Blend radius: 0 mm
Bodies: Implicit Body List (2) — Implicit Body list_0
0: Implicit Body_1
1: TPMS — Implicit Body_4
Body: Implicit Body_2
Cell size: 2 2 2 mm
Approx. bias length: bias_length_4 mm
Fill type: Gyroid
Feature size: 0.25 mm
Adaptivity: 0.3

Export Mesh — Mesh File Data_0
Path: C:/Users/Morten/OneDriv
Mesh: Mesh from Implicit Body — Mesh_0
Body: Boolean Union — Implicit Body_8
Blend type: Continuous
Blend radius: 0 mm
Bodies: Implicit Body List (2) — Implicit Body list_0
0: Implicit Body_1
1: TPMS — Implicit Body_4
Body: Implicit Body_2
Cell size: 2 2 2 mm
Approx. bias length: bias_length_4 mm
Fill type: Gyroid
Feature size: 0.25 mm
Adaptivity: 0.3
Units: mm

Figure 65: Showing how the building blocks for a nTopology geometry is created

Results for Lattice structures

The original plan was to 3d print the lattice structure specimens; however, in order to complete the thesis in time, FEA was chosen. Several attempts were made to conduct the FEA with the exact same geometry as nTopology produced, but along the way, the complexity of the mesh made the analysis in Abaqus extraordinarily complex. Therefore, the area with the lattice structure was replaced with a void. The effect of the simplification will be outlined in the discussion chapter. The FEA was done in the same manner as already described in "Test Procedure FEA".

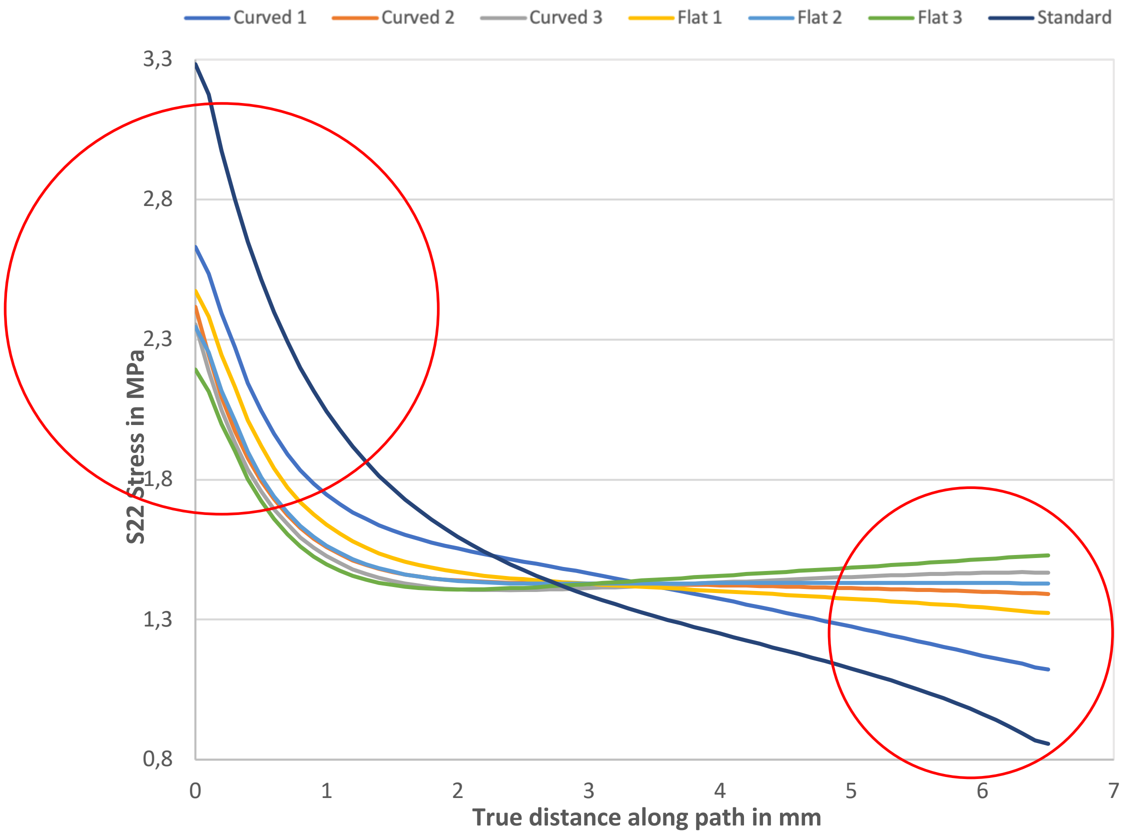


Figure 66: Stress along path in mm. Red circles show areas that will be zoomed in further

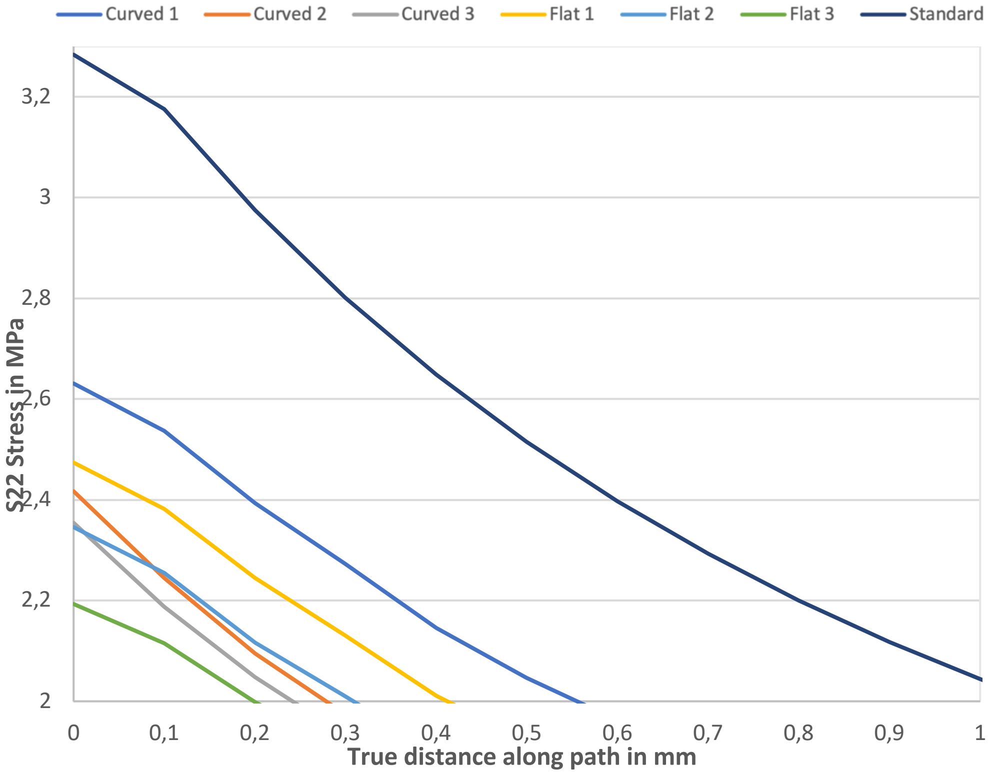


Figure 67: Magnified view of stress along path for section 0-1 located closest to the notch

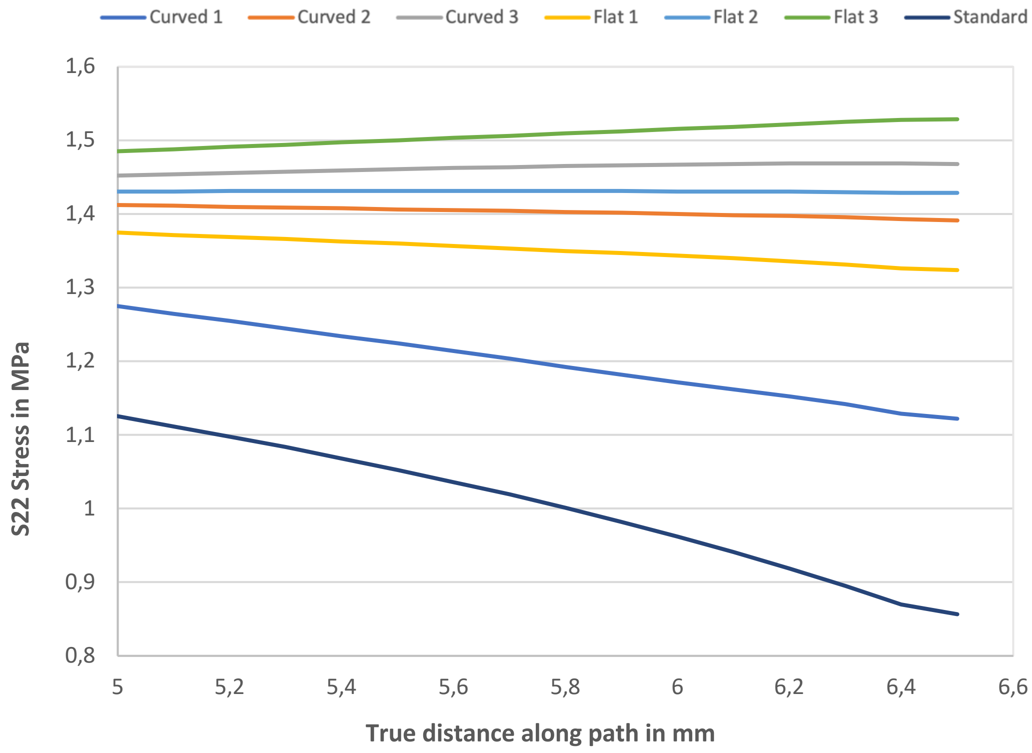


Figure 68: Magnified view of stress along path for section 5-6.5 located furthest away from the notch

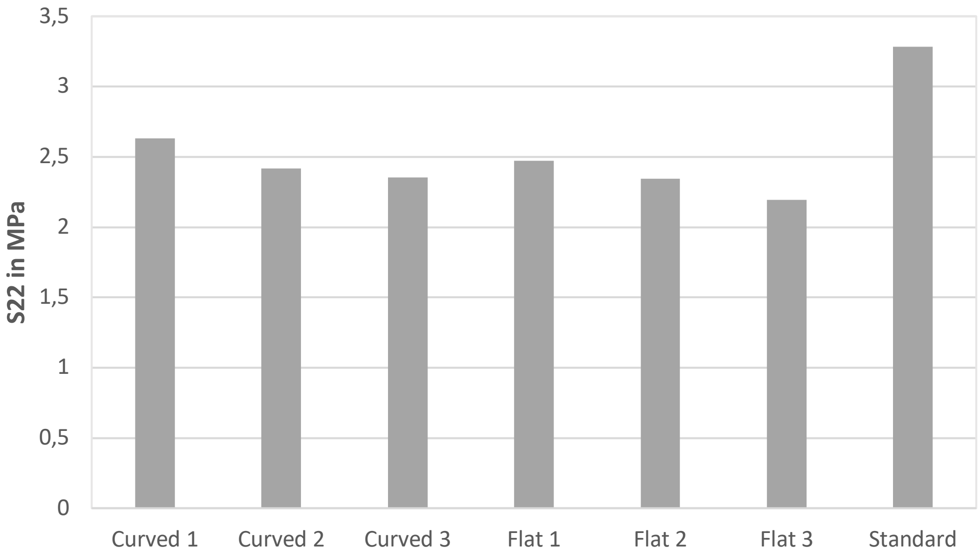


Figure 69: Bar plot showing max S22 stress in MPa at center hole for each case

Discussion

A new method of adding tailored voids in a specimen was proposed for reducing concentrations of stress in the specimen.

Initial FEM Analysis

Examining Figure 16 gives away certain traits of the different geometries, radiuses and distances. Case 1 in the top left corner had the highest stress out of all the cases at 291.8 MPa, shown in Table 1.

Looking into the size-effect of the holes, it also becomes clear that the larger the added geometries, the lower the stress in the center hole. For example, by looking at Case 4 with distance fixed at 2 mm, the highest stress went from 275.3 in Case 4.1.1, to 245.1 in Case 4.2.1, to 210.0 in Case 4.3.1 when Radius 2 went from 5mm, to 7.5mm to 10mm. The same behavior can be observed for all the cases. However, with regards to the largest diameter for each case, there was a higher stress appearing in the added geometry than what was observed in the center hole, which defeats the purpose. This tells us that the ideal size would be somewhere between Case X.2.X and Case X.3.X, which would give the same stress in the center hole and the support hole, when the goal is to have the lowest appearing S22 stress.

Looking into the distance-effect of the charts does not give us the same uniform response as that of the size-effect, but there are some interesting takeaways. Case 5 has the most unique geometry and will be commented on later. The two smallest radiuses for Cases 2, 3 and 4, Radius 1 = 5mm and 7.5mm, show that all stress in the center hole are slightly lower when increasing v1. When the stress is higher in the support holes than in the center holes, at Radius 1 = 10 mm, the stress increases as v1 increases.

Looking at both graphs for Case 5, it is clear that the stress increases with distance v1. This may come down to the fact that some of the stress gets channeled in between the two holes next to each other, as seen in Figure 27. If v1 increases, the channeled stress has more material to escape through around the center hole.

The reduction in the stresses around the center hole still has to travel through the specimen somehow. Figures 11-20 visualize this issue well. The graphs have some variation in shape, but the effect of the larger radius is evident. When the stress at the center hole decreases, the stress shifts to the right, and into the lesser stressed part of the specimen. The difference along the distance variable is so small, that the graps lay almost on top of each other. The difference along the radius variable is, however, large, and easy to observe. For all the cases, the graphs intersect at about 12mm, which is an interesting observation.

Looking at the contour plots gives a good idea of how the stresses are distributed throughout the specimen. There are more similarities than differences across the cases. On both sides of the center hole, the positive S22 data forms a butterfly-like geometry. Above and below the center hole, the material is compressed and gives negative S22 values.

Fatigue analysis

The experimental fatigue analysis only had 5 geometries, down from the initial 34 unique geometries in the initial FEM analysis. After deciding the final 5 geometries, a new FEA was conducted. The maximum S22 obtained can be seen in Figure 36. The results seen were in line with what had already been observed in the initial FEM analysis. The best performing geometry was Case 5 with the largest radius and a smaller hole on top and below, at 441.4 in MPa. However, it performed only slightly better than Case 3 at 449.6 MPa. Cases 2 and 4 were also close, at 491.4 MPa and 489.9 MPa respectively. Case 1, with no holes, had a stress of 511.9 MPa.

Looking at Figure 35, the contour plot can be observed. There are a lot of similarities between Figure 25 and Figure 31. In Figure 31, it becomes even clearer how the stress travelling from the top and the bottom does not go straight to the center hole. Instead, it is manipulated into the lesser stressed area. The "butterfly wings" get larger, and the darkest spots close to the center hole get lighter.

In figures 37 and 38, repetitions until failure can be observed. The best performing specimen is Case 5.3 at 86 047. The best performing specimen for Case 1 was at 42 199. The

improvement in Case 5 is 103.9%. With a small sample size, there will be some variation in the results. The difference between Cases 3 and 5 is not significant, and might indicate that the smallest hole in Case 5 could be avoided. Cases 2 and 4 did not significantly impact fatigue life, since the holes are too small to do a difference in redistributing the stress. Unfortunately, the surface finish for case 4 was coarser than the rest, which resulted in premature failure. One would expect that the specimens would have performed slightly better than Case 2 with good surface finish.
It was shown that the fatigue life improvement in the proposed geometries significantly reduced stresses, and improved fatigue life. Although the results were obtained for aluminum, the same approach can be employed to study the fatigue life improvement in other metallic alloys and other types of specimens. The new geometries proposed for stress redistribution can be recommended as an effective way of extending fatigue life in metallic structures.

Second FEA Stress Analysis

Looking at Figure 44, it is clear that the two outliers for the stress in the added notches are Notch-5-1 and 7-1. The stress is even higher in the added notch than in the main notch. This makes sense, since Notch-5-1 and 7-1 also have slightly lower stress in the center hole than the rest, which can be observed in Figure 42.

The rest of the specimens specimens do not vary as much as the two outliers. As expected, Notch-1 without added tailored voids experiences the highest stress, whereas the plain specimen without any holes has the lowest stress.

Looking closely at Figure 46 and 47 reveals that the specimens with the highest stress closest to the center tend to have the lowest further away. However, there are a couple of exceptions, namely 7-1 with the lowest center stress would be expected to have the highest stress all the way to the right edge, but does in fact, only have the fifth highest stress.

Varying the distance between the center hole and the tailored void seems to affect the S22 stress the most when the void is further out from the center. Notch 3-1, 3-2 and 3-3 have almost the exact same maximum stress at the center, while 7-1 and 7-2 have much larger differences in stress.

Second Fatigue analysis

For the second fatigue analysis, one would expect larger differences in fatigue life between specimens than for the FEA. The fatigue life was shown for both 200 MPa and 150 MPa in Figure 53, and separately in Figure 54 and Figure 55 for better readability.

Notch-1 without tailored voids was clearly the weakest specimen for both 150 and 200 MPa. The best performing single case specimen for 150 MPa was Notch-5-1 with a solid 72% increase over the best performing case for Notch-1. Notch 7-1 came close with its best performing single case 70% higher.

All specimens failed from a crack growing from the center hole eventually separating the specimen. This is really interesting, because one might expect that for Notch 5-2 and 7-2, the failure would happen in the added tailored void, since that is where the highest stress in the FEA was located. This might indicate that higher stress in smaller voids is not as critical as in larger voids. For 200 MPa, Both 5-1 and 7-1 performed better than 5-2 and 7-2 with the voids larger away from the center. For Notch 3-1, 3-2 and 3-3, the average fatigue life was surprisingly similar with 27.7K, 26.1K and 26.1K.

Once the number of cycles increases and the stress is dropped down to 150 MPa, the best performing notches change. Notch 7-2 has the best performing single case with an increase in fatigue life of 200% over the best performing case for Notch-1. It is difficult to draw certain conclusions from the fatigue analysis without larger sample sizes, but the magnitude of the effect from redistributing the stress in the specimens is clear. The worst performing single case specimen for 150 MPa is Notch 3-2 with a 56% increase in fatigue life.

Lattice structures

The specimen named “Standard” is the dogbone with a center hole, and no added tailored voids. The FEA results from the lattice structured specimens deliver the best performance by far. The oversized tailored voids impressively utilize the lesser stressed parts of the specimens.

For the curved specimens, a singularity for the S22 stress appears in the sharp edges of the half moon that increases with finer mesh. This stress was neglected, and expected to disappear if the lattice structure was included in the FEA.

The clear trend is that the larger the void, and the further away the void is, the lower the S22 stress. Flat 3 has the lowest S22 stress, 32,3% lower than the Standard specimen. If a fatigue experiment would follow the same trend as experienced earlier, the fatigue life would also drastically increase. With the lattice structures added, some of the effect would slightly decrease, but the lattice would improve strength for mixed axial loading and compression.

References

[1]
Suresh, S. (2004). *Fatigue of Materials*. Cambridge University Press. ISBN 978-0-521-57046-6.

[2]
Changes in the Fatigue Life of Plates with Attachments Due to Geometrical Effects" was published. I. F. C. Smith and T. R. Gurney
http://files.aws.org/wj/supplement/WJ_1986_09_s244.pdf

[3]
A Method for Predicting the Effects of Specimen Geometry and Loading Condition on Fatigue Strength
Chengqi Sun and Qingyuan Song
https://www.mdpi.com/2075-4701/8/10/811

[4]
https://www.researchgate.net/figure/Bugatti-brake-caliper-currently-the-worlds-largest-functional-part-produced-in-titanium_fig5_332092139

[5]
Course work TMM4165