TR125513

Capitalizing on Additive Manufacturing Using Autodesk Generative Design

Daniel Noviello
Autodesk Advanced Consulting

Learning Objectives

- Learn how to set up starting and obstacle geometry for Autodesk Generative Design inside Fusion 360
- Learn how to use Autodesk Generative Design to re-design a structural aerospace bracket
- Learn how to use Fusion 360 non-linear finite element simulation to verify additive manufactured designs
- Understand the potential of generatively designed AM parts compared against typical subtractive (machined) parts

Description

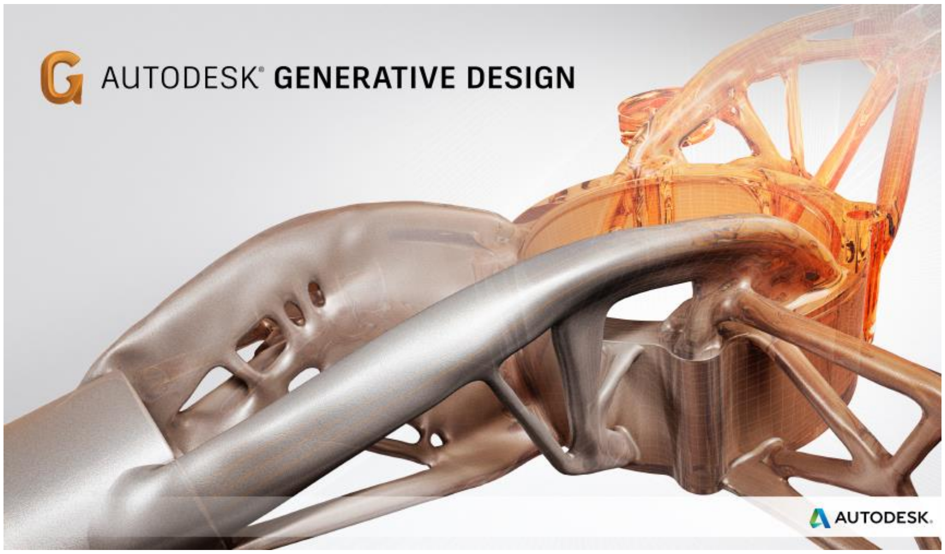
With the announcement of Autodesk Generative Design, we take one of the most common structural engineering problems and turn it on its head. This class will teach you how to use AGD to discover unthought-of designs for a structural aerospace bracket. Learn how to go from seed geometry definition in Fusion 360, to problem definition, design generation, and results exploration in Autodesk Generative Design. Select from generated designs of different materials, safety factors and build constraints. Next, we use detailed non-linear finite element analysis in Fusion 360 to verify the selected Generative Design solution. Learn how to input non-linear properties for AM metals and correctly interpret finite element results. Once verified we will discuss AM (metal powder bed) build preparation with the help of Autodesk Netfabb. Finally, we will compare the performance of our new Generatively Designed bracket with that of the baseline (subtractive manufactured) bracket by physically breaking both of them!

Speaker(s)

Daniel Noviello is a Principal Technical Consultant in the Autodesk Advanced Consulting Team specializing in additive manufacturing, and structural design, analysis and optimization. He graduated from University of Queensland, Australia with a First-Class Honors in Mechanical and Space Engineering. Having worked for various aerospace / space companies, including GKN and Surrey Satellite Technology Limited, Daniel brings key industry experience to the team. He has a passion for the practical application of new and emerging technologies to help advance the design and manufacturing industry.

Introduction to Autodesk Generative Design

Autodesk Generative Design is a new product that allows designers and engineers to rapidly generate multiple designs based on a set of inputs describing the physical requirements. Unlike traditional optimization tools, AGD uses an open design space to provide multiple solutions which compete to meet specified objectives. This means the user can view the full solution space with potentially hundreds of designs and easily determine trade-offs between performance parameters like mass, stiffness and strength, cost, and manufacturability.



The Generative Design Workflow

Consider the traditional design cycle; an engineering team takes a set of requirements like loads, constraints and form. It then designs a nominal structure that may or may not satisfy the requirements. This nominal structure is analyzed and then redesigned over several iterations until it meets the original requirements. This process takes a long time and is particularly laborious when requirements change mid-process.

In the generative design workflow, the user inputs all requirements into the software. They also input design options like different materials, build orientations and safety factors. This allows the software to generate many designs that are fully analyzed. Instead of having to iterate and redesign, the engineer can navigate a range of designs for which all requirements have been met. Should requirements or specifications change, they can simply choose another design from the solution space.

Objectives

The GrabCAD Alcoa Bracket Challenge

The GrabCAD Alcoa bracket challenge was held in 2016 and presented an airplane bearing bracket for redesign in order to minimize mass while meeting a range of technical requirements. It was defined here https://grabcad.com/challenges/airplane-bearing-bracket-challenge as follows:

The objective of this challenge is to redesign the bearing bracket in such a way that its topology and shape are optimized for minimizing weight while fitting in the target envelope and meeting the technical requirements. The bracket is intended to be additively manufactured and the design shall also minimize and/or eliminate the need for support structures. The submitted designs will be evaluated via FEA and ranked based on their strength-to-weight ratio. The top five designs will be fabricated via additive manufacturing and tested. The winners will be selected based on mechanical performance and on the cost associated with the additional manufacturing operations to remove support structures.

A total of 303 different entries were received, each using different design, manufacturing, and optimization techniques.

Requirements

Original requirements as per https://grabcad.com/challenges/airplane-bearing-bracket-challenge were:

- The design must fit entirely within target envelope described in the specifications.

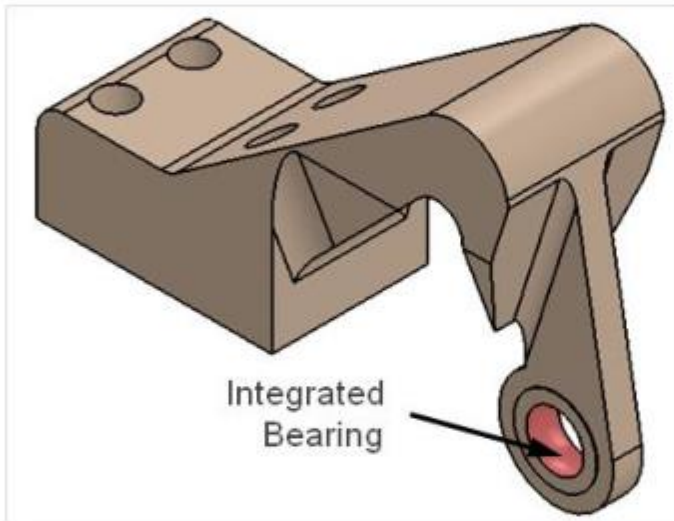
- Design material: 15-5PH per AMS5862
  - Elastic Modulus (E) = 29,000 ksi = 200,000 MPa = 200 GPa
  - Poisson Ratio (v) = 0.27
  - Yield Stress (σ_y) = 145 ksi = 1000 MPa
  - Density (ρ) = 0.283 lb/in3 = 7833 kg/m^3
  - Material is assumed to be linear elastic

- Minimum geometric feature: 0.025 in.

- Minimum wall thickness: 0.045 in.

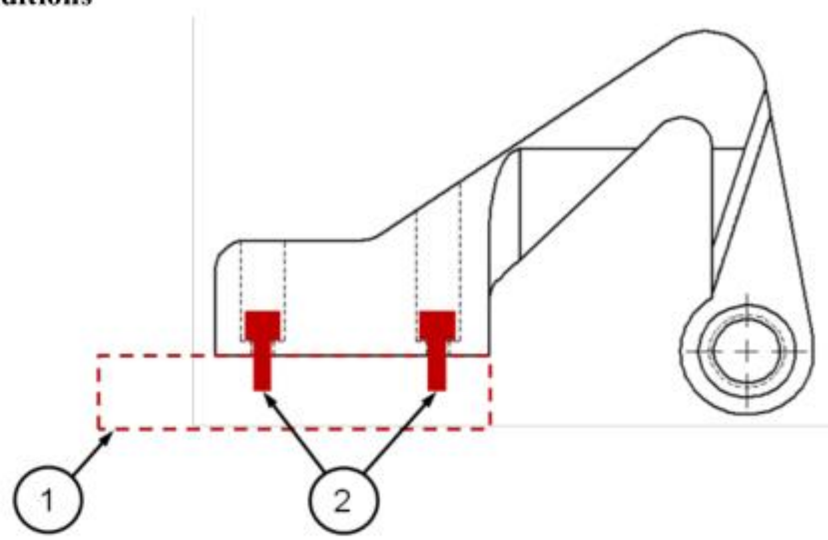
- Parts shall be optimized for minimum weight with the following boundary and loading conditions:
  - Base support: The part is bolted against a mating plate of high stiffness
  - Bolts interface: The parts is fastened with four #10-32 high strength tension rated bolts as indicated in the specifications
  - Bearing interface: The part is loaded through a high stiffness spherical bearing with three load cases:
    1. A load of 1,250 lbf applied horizontally
    2. A load of 1,875 lbf applied 45 degrees from the horizontal
    3. A load of 2,500 lbf applied vertically

Challenge Requirements and Work Envelope



Part work envelope available for download in STEP format

Boundary Conditions

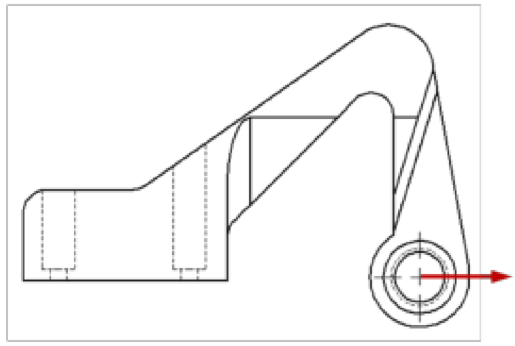


1. Part is simply supported by a stiff plate
2. Part is fastened by four high strength bolts (#10-32)

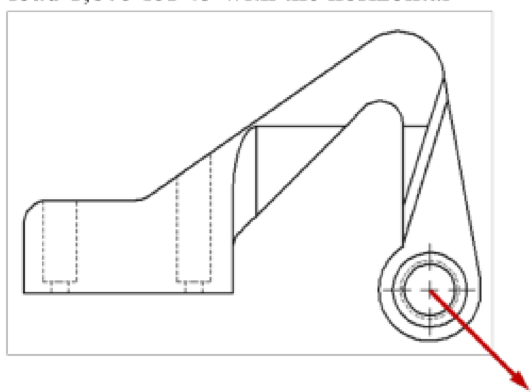
Figure 1 - GrabCAD Challenge Definition (1) [source: grabcad.com]

Loading Conditions

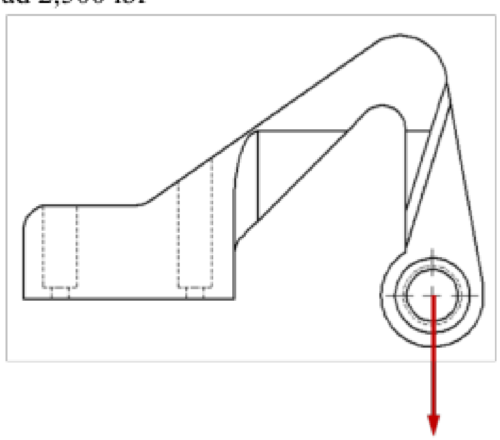
Load case 1: Horizontal load 1,250 lbf



Load case 2: Inclined load 1,875 lbf 45 with the horizontal



Load case 3: Vertical load 2,500 lbf



In all load cases, the loading shall be applied statically, through a stiff spherical bearing with .3125in in diameter.

Figure 2 - GrabCAD Challenge Definition (2) [source: grabcad.com]

Objectives of This Study

This study aims to demonstrate the capability of Autodesk Generative Design in the context of the GrabCAD challenge. Specifically:

1. Redesign the bracket with a requirements-only specification in AGD. This should satisfy all load cases, material options and manufacturing requirements.

2. Demonstrate a solution space with various mass-minimized designs.

3. Verify the chosen AGD-generated design using a detailed finite element model.

4. Prove AGD-generated designs by manufacture.

5. Validate the design by physically testing it.

Adapting Requirements for the Current Study

For the purposes of demonstration and cost-effectiveness, a few requirements were adjusted for this study.

Material

The central change was the material from 15-5PH steel to 2014 T651 aluminum. Precipitation hardened steel (15-5PH per AMS5862) is exceptionally strong and stiff and more expensive. It would also make physical testing more difficult as a larger load cell would be required together with stronger support frames and jig equipment. By selecting aluminum, the test is far more economical, and minimizes the risk of equipment failure.

Loads

With the change of material, the bracket will expectedly perform much worse under the original prescribed loads. To make the loads more suitable for the new material, they were scaled down by the ratio of the aluminum yield strength to the steel yield strength.

Yield strength of steel*:	1000 MPa
Yield strength of 2014 T651 aluminum**:	447 MPa
Load reduction factor:	447 / 1000 = 0.447

New loads:

1. **Case 1 (0° orientation):** 0.447 x 1250 lbf = 558.8 lbf = **2485 N**
2. **Case 2 (45° orientation):** 0.447 x 1875 lbf = 838.1 lbf = **3728 N**
3. **Case 3 (90° orientation):** 0.447 x 2500 lbf = 1117 lbf = **4971 N**

* *Taken from original GrabCAD problem definition*
** *Specification property for stock material used to make original machined bracket that was used in this physical test.*

Space envelope

The space envelope was relaxed from the original requirement of being completely within the existing bracket. The new space envelope includes the original bracket plus the yellow space indicated below.

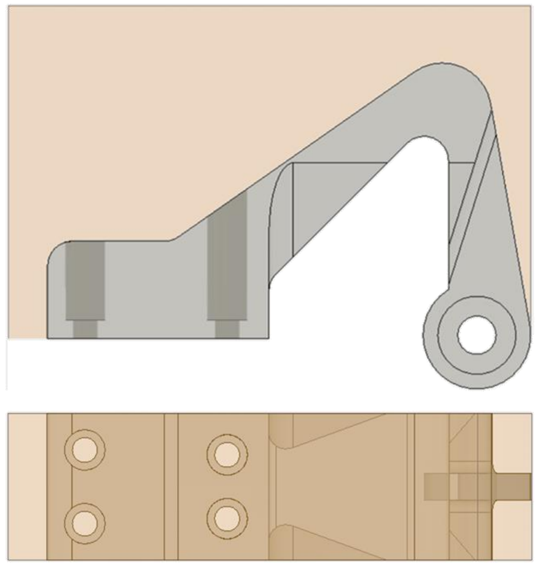


Figure 3 - Adapted space envelope for generative design

Generative design goes beyond component optimization and for this reason, it is better to allow enough space for various designs to be fully explored.

Spherical Bearing

Since the test part will not be used in service, the spherical bearing in the original specification was omitted. Instead, a simple hole of 10mm diameter was used to load the bracket.

Setting Up Starting and Obstacle Geometry using Fusion 360

Along with a few other inputs, Autodesk Generative Design requires a definition of 3 categories of geometry:

1. Starting geometry (optional)
2. Preserved regions
3. Obstacles (optional)

This geometry can be imported from IGES, OBJ, STL, SAT or STEP formats. In this example, the design space is defined in Fusion 360 and exported as a SAT file with multiple bodies.

Sketching

The various bodies are generated by first importing the original bracket into Fusion 360. At this point, the outlining edges of the geometry can be projected onto an appropriate plane and the resultant profile extruded.

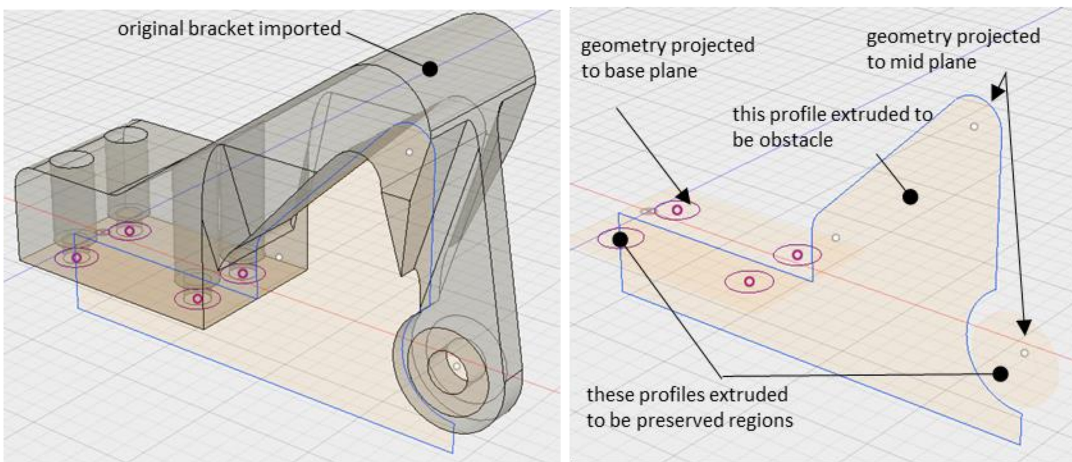


Figure 4 - Imported geometry with example profiles (left), example profiles for selected preserved regions and obstacles (right)

Preserved Regions

Using a series of basic profile extrusions in Fusion 360, the following preserved regions are created as separate bodies.

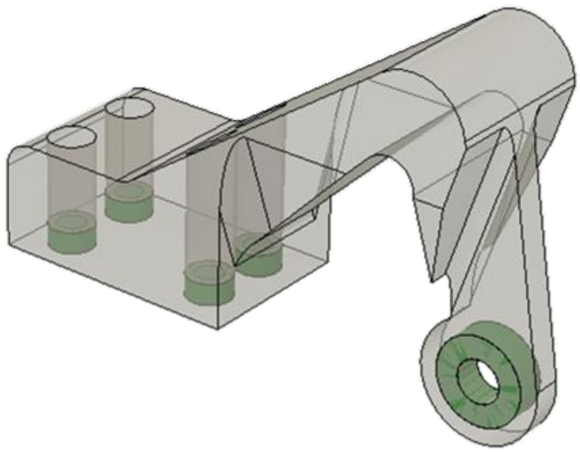


Figure 5 - Original bracket with preserved region bodies (green)

It is important to preserve any important interfaces, especially those that need to have flat surfaces and sharp edges.

Obstacles

There are a number of obstacles required for this problem. These are largely due to the function of the part. Firstly, space is needed to reach and tighten the fastener heads as modelled by the tall cylinders (a). Next the attaching surface will be present and it is assumed that the space under the bend in the bracket must be unoccupied (b). Lastly, there must be space for a load pin and structure used to apply the load (c).

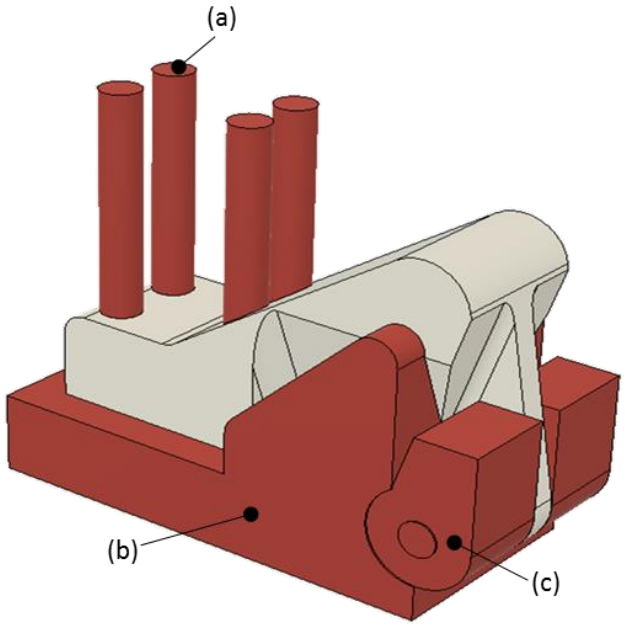


Figure 6 - Original bracket with obstacle bodies (red)

Starting Geometry

While not completely necessary, specifying starting geometry helps produce designs that have a more predictable outcome. In the same way, complex starting geometry can inspire designs that may be counter-intuitive to the user, but satisfy the requirements nonetheless.

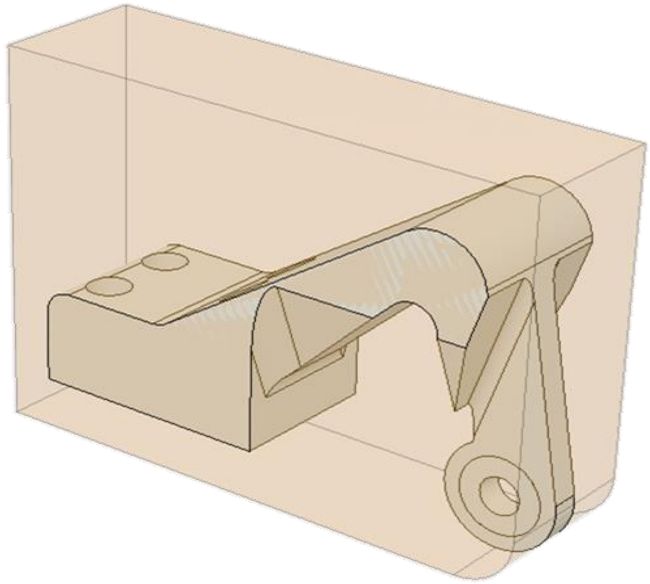


Figure 7 - Original bracket with starting geometry (yellow)

Exporting Geometry from Fusion 360

Once the various bodies are defined, they are made visible and exported in SAT format ready for Autodesk Generative Design.

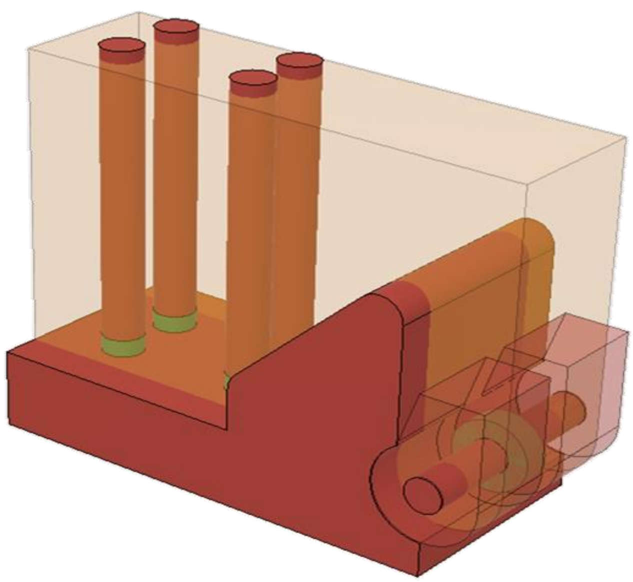


Figure 8 - Preserved regions, obstacles and starting geometry exported to SAT file

Generate Solutions using Autodesk Generative Design

This section provides an overview of the process required to redesign the GrabCAD challenge bracket using Autodesk Generative Design (AGD).

Importing Geometry

In this example, all geometry is imported using a single SAT file. It is important to note however, that multiple geometry files can be imported from different sources. This can be useful for particularly complex arrangements of obstacles / preserved regions. For example, one could use three separate SAT files, one for each of obstacles / preserved regions / starting geometry.

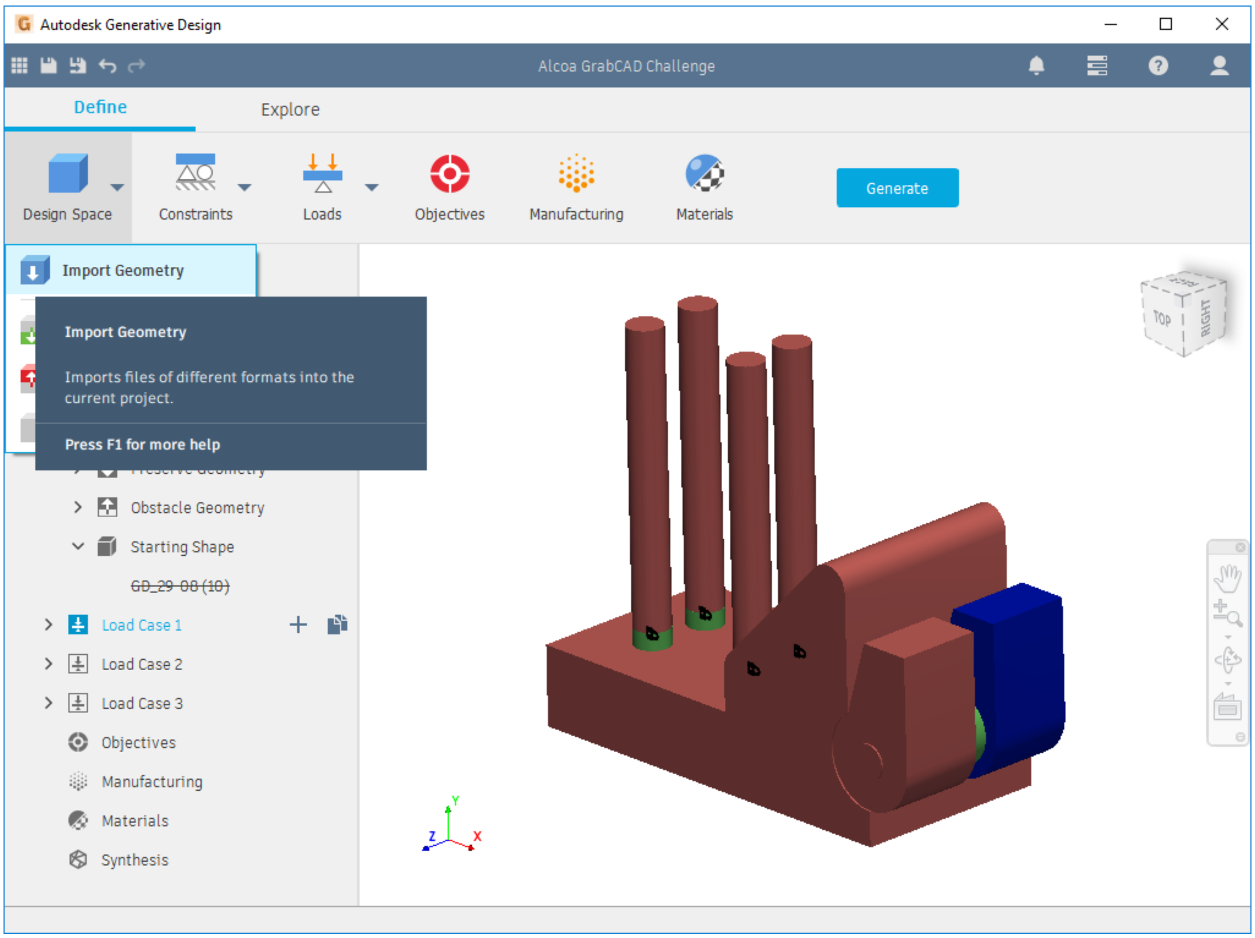


Figure 9 - Importing Geometry into Autodesk Generative Design

Assigning Geometry

The next step in AGD is to assign each body to be a preserved region, obstacle, or starting geometry as applicable. This can be done via context menus on the geometry objects in the tree, directly on the objects in the viewport, or using the 'Design Space' menu.

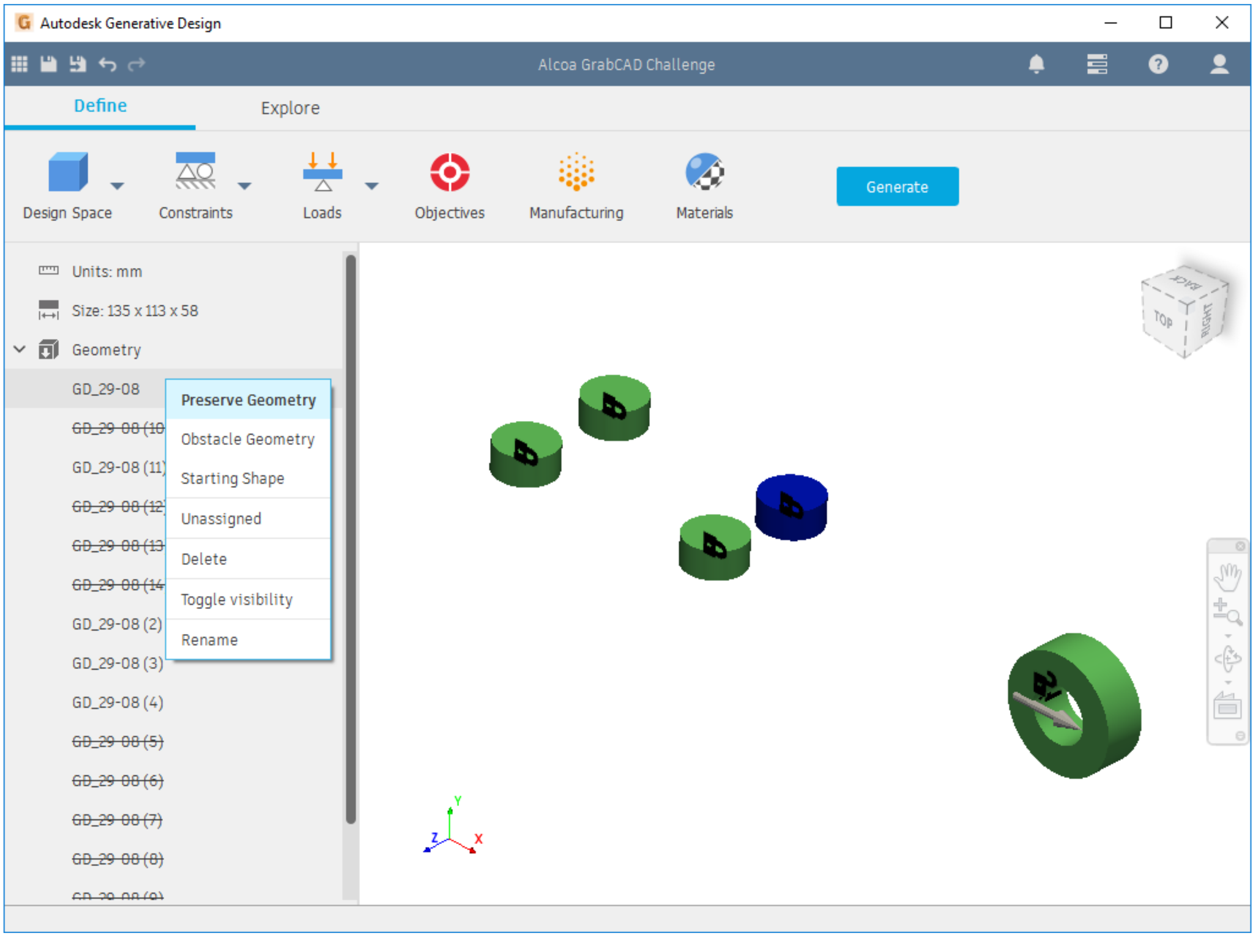


Figure 10 - Assigning a preserved region in the tree

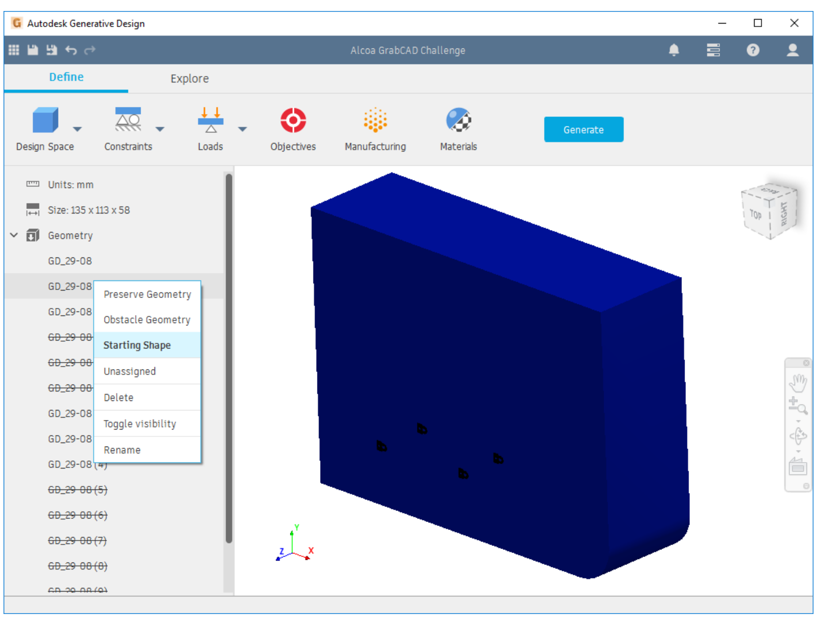


Figure 11 - Assigning starting geometry in the tree

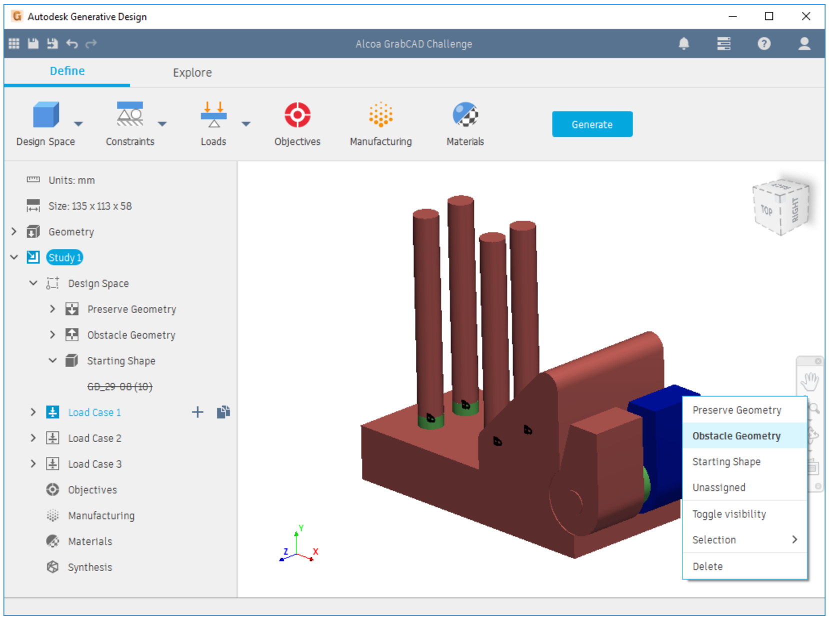


Figure 12 - Assigning an obstacle directly in the viewport

Once assigned, bodies of each preserved region, obstacle and starting geometry are automatically colored green, red or yellow respectively.

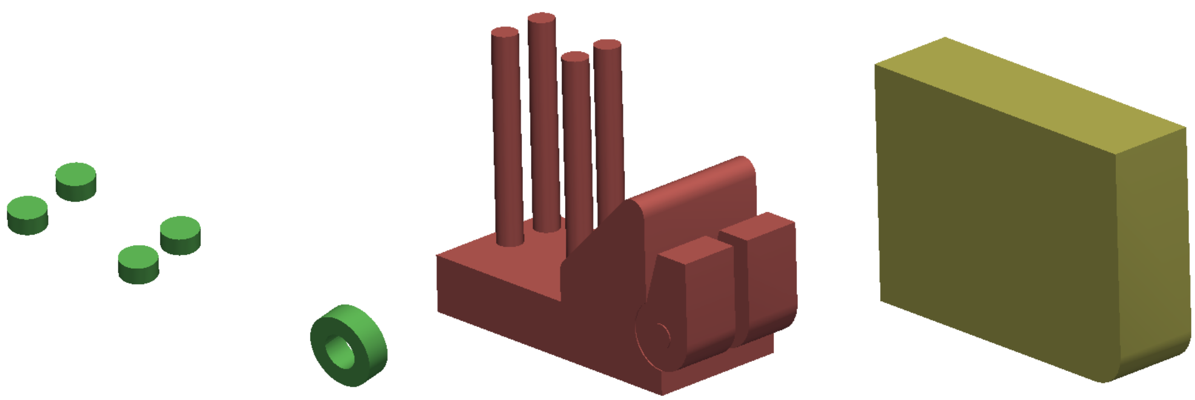


Figure 13 - Colors of preserved regions, obstacles and starting geometry (in order from left to right)

Defining Loads and Constraints

The next step in defining the problem is to add the loads and constraints in as many load cases as required. For this problem, three separate load cases are added using the tree context menu.

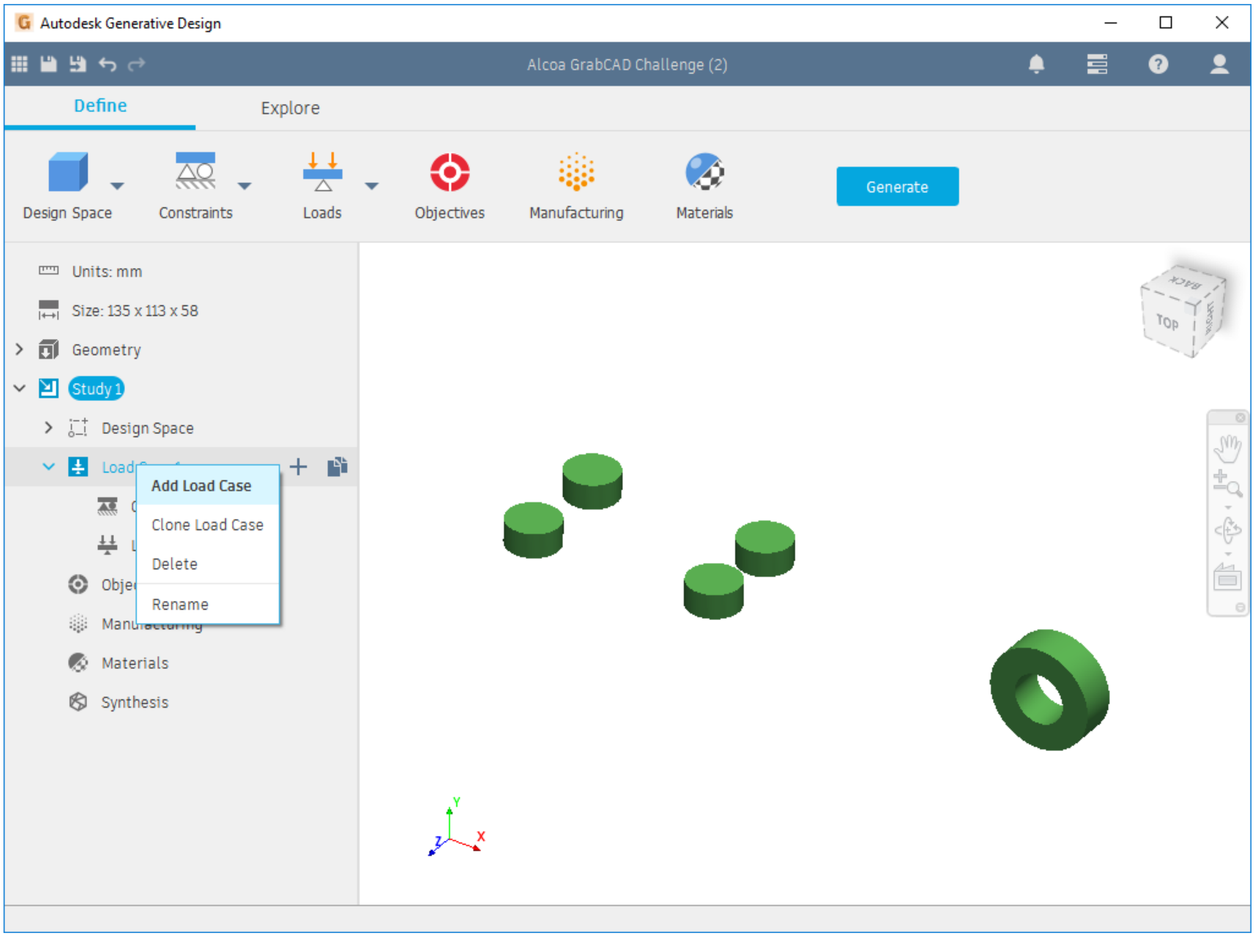


Figure 14 - Adding a load case using the tree

Subsequently, loads and constraints are added via the top menu. Loads and constraints can be applied to any face, edge, vertex or entire body. The following images show body constraints on the four attaching bolt preserved regions (in X, Y, Z), face constraints on the flat sides of the load lug (in Z), and a force applied to the inner diameter of the lug.

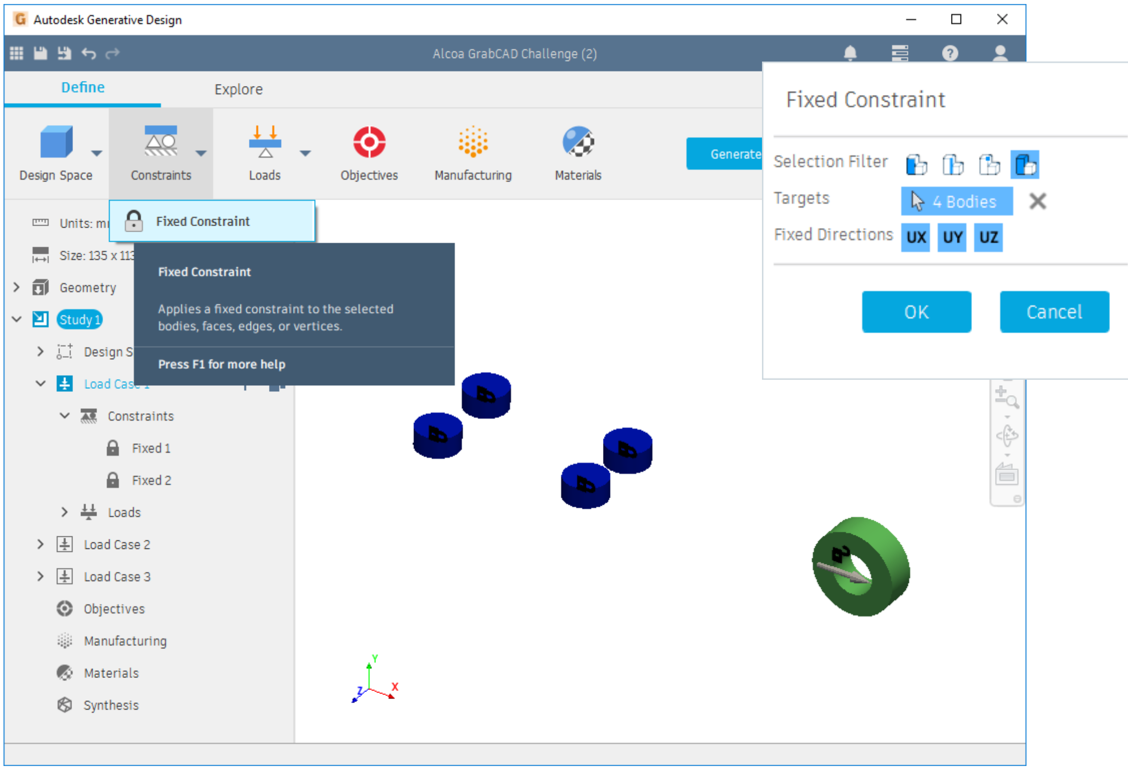


Figure 15 - Defining a constraint using the top menu

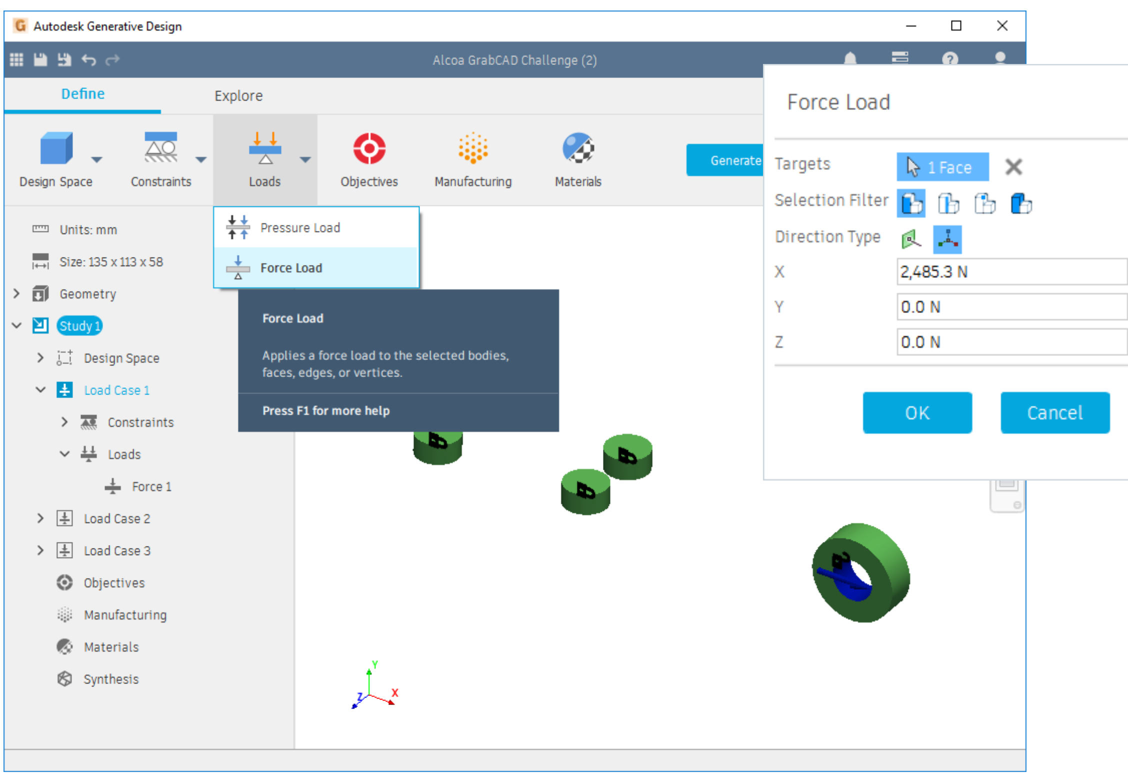


Figure 16 - Defining a force using the top menu

Specifying Objectives and Manufacturing Constraints

Autodesk Generative Design is then given the overall objective for each design; minimize mass. A range of acceptable safety factors is also provided, this is based on the exceedance of the material yield strength. Other objectives include: mass target, whereby resultant designs aim to meet a specified mass, and maximize stiffness where stiffness will be maximized within the given safety factor (stress) limits.

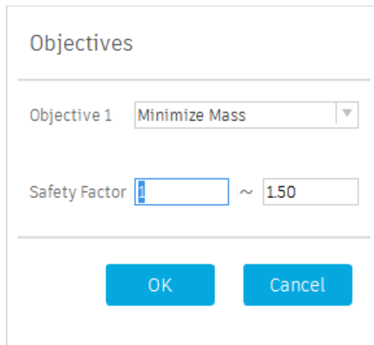


Figure 17 - Minimize mass objective

Along with objectives, manufacturing constraints are input. For the current problem, only additive manufacturing (metal powder bed) is considered, and this is for both an unrestricted build, and a +Y directional build. This implies that +Y is the vertical direction inside the print chamber.

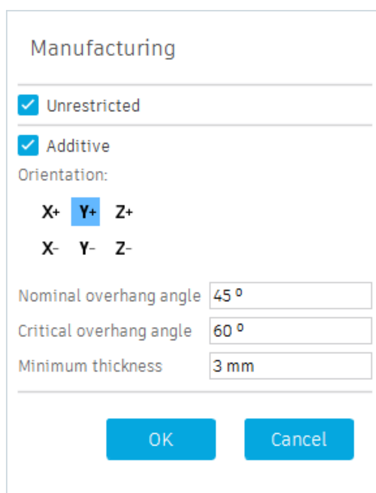


Figure 18 - Manufacturing constraints

Other manufacturing parameters include the following:

- **Nominal overhang angle**:
  the target allowable angle that the as-built material shall make with the vertical plane

- **Critical overhang angle**:
  the maximum allowable overhang angle

- **Minimum thickness**:
  the minimum face-to-face thickness throughout the design

Specifying Material Options

The final input required by AGD is a range of permissible materials. Choosing multiple materials will facilitate a large design space however in this problem, since the material is predetermined, a single material is selected (AlSi10Mg).

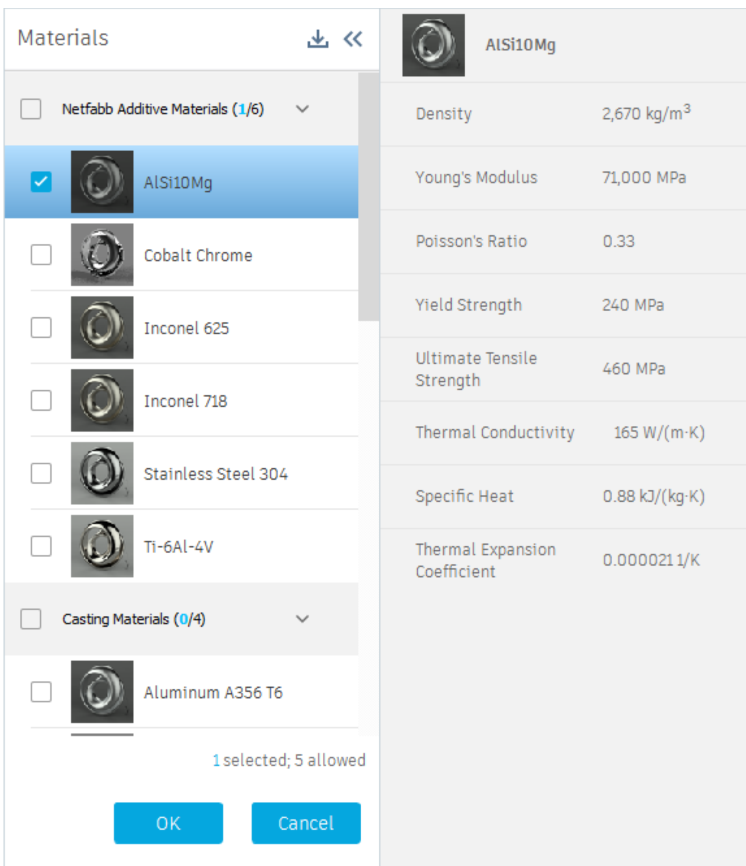


Figure 19 - Material specification

Generate and Explore!

When the user clicks 'Generate' the problem is submitted to the cloud and the designs begin to generate. Designs are immediately visible while they process in the 'Explore' tab.

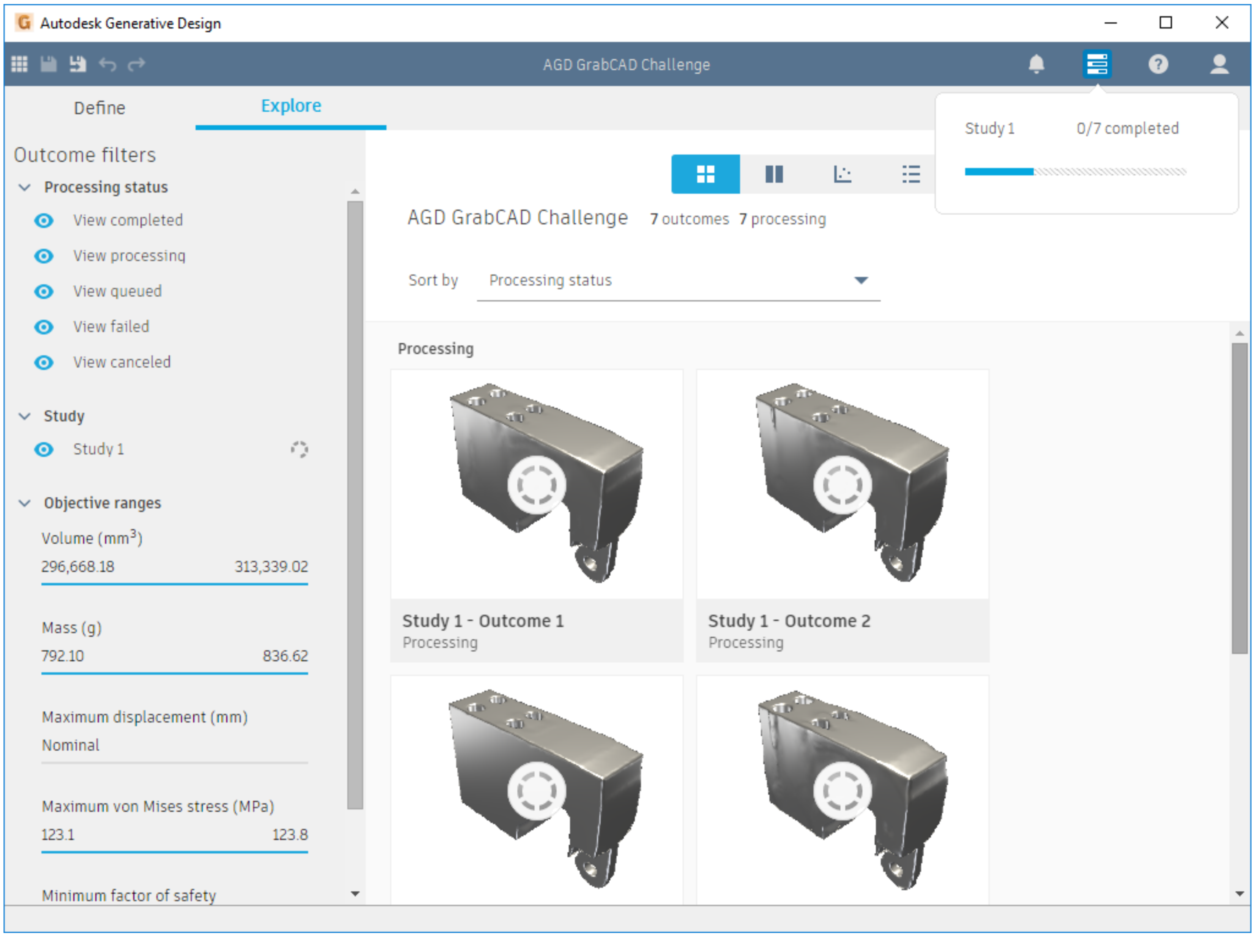


Figure 20 - Processing designs in the Explore Tab

The resultant designs can be visualized in various ways. The image below shows a plot of maximum displacement versus mass with additional details of the selected design.

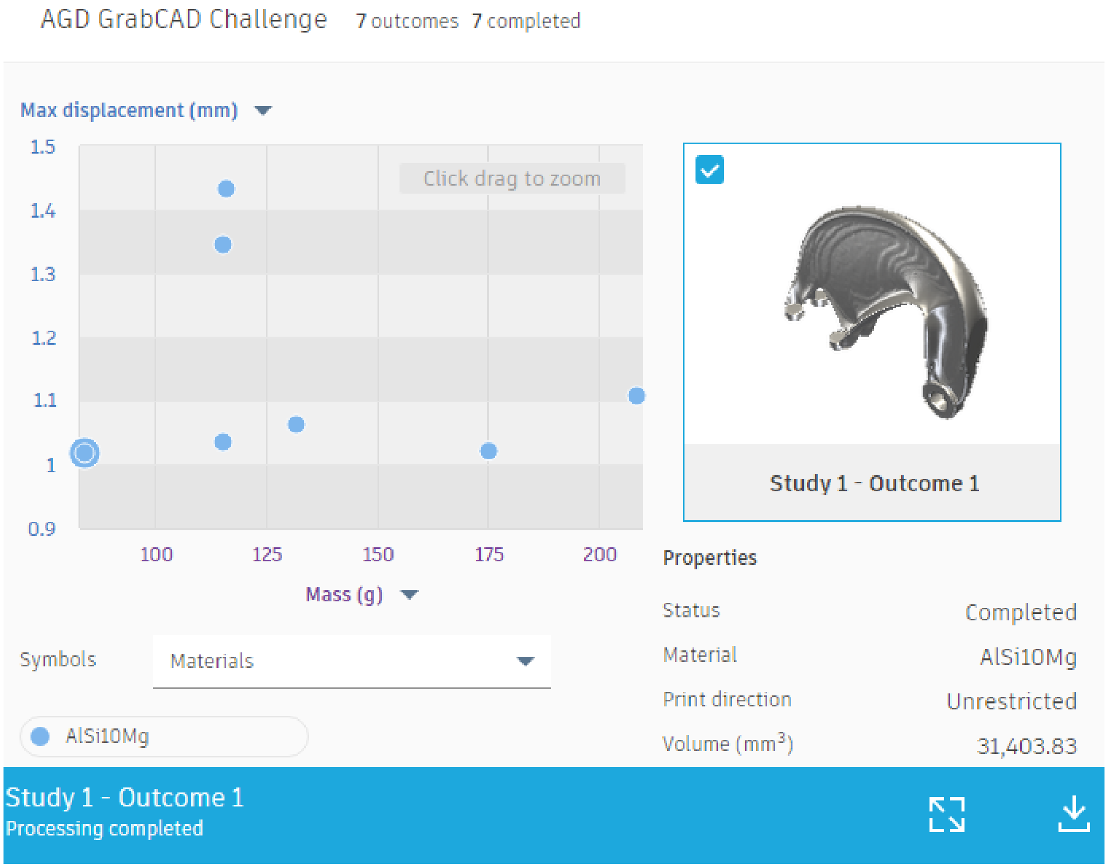


Figure 21 - Navigating the solution space

After being converted into a boundary representation CAD format, the resultant models were overlaid in Fusion 360 to display the conformance of the final selected geometries.

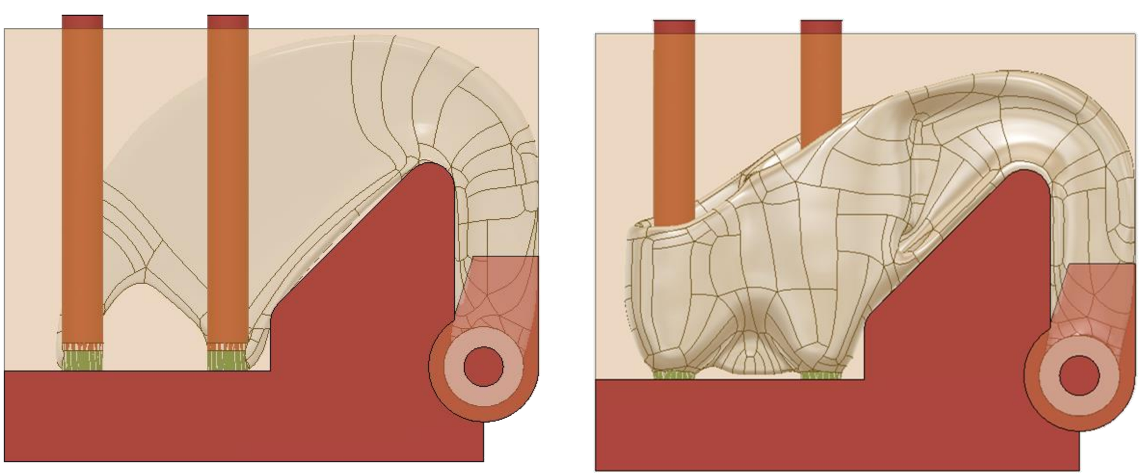


Figure 22 - Unrestricted build (left) and +Y build (right) selected solutions overlaid in starting geometry

Nastran Verification with Fusion 360

A note on Boundary Representation Conversion

When converting to BREP, the mesh representation in AGD is slightly modified due to expected deviations when reconstructing the surfaces. Additionally, there are specific modifications made to the interfaces, i.e. reinstating of sharp edges. These modifications result in expected differences of the verification results to those computed inside AGD.

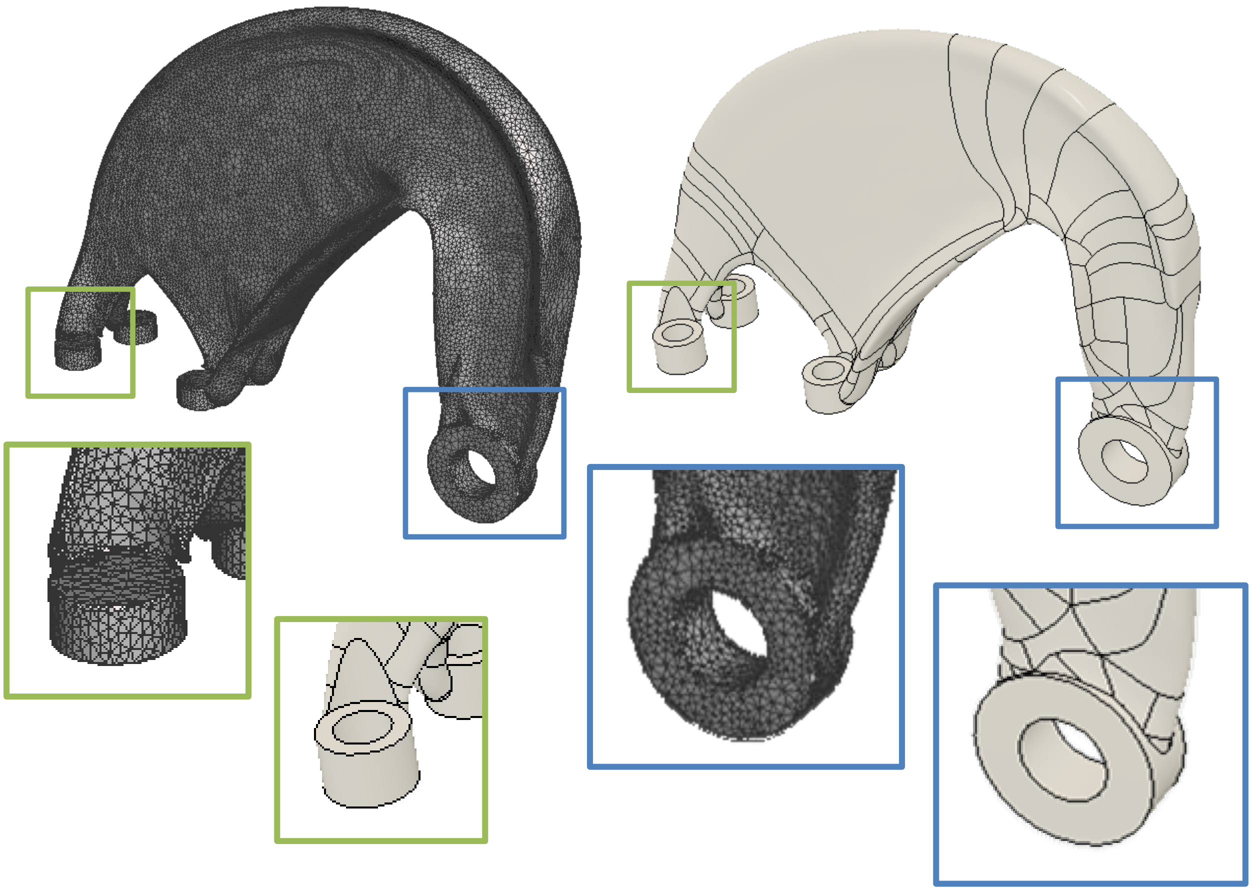


Figure 23 - Differences of the BREP CAD model to the AGD mesh

Setting Up a Linear Static Solution

The linear static solution can be accessed via the Fusion 360 Simulation Environment. A Static Stress study is created and the three load cases modelled in the same way as done in AGD. The solution can be solved locally or on the cloud, and then results are returned and interrogated by the user.

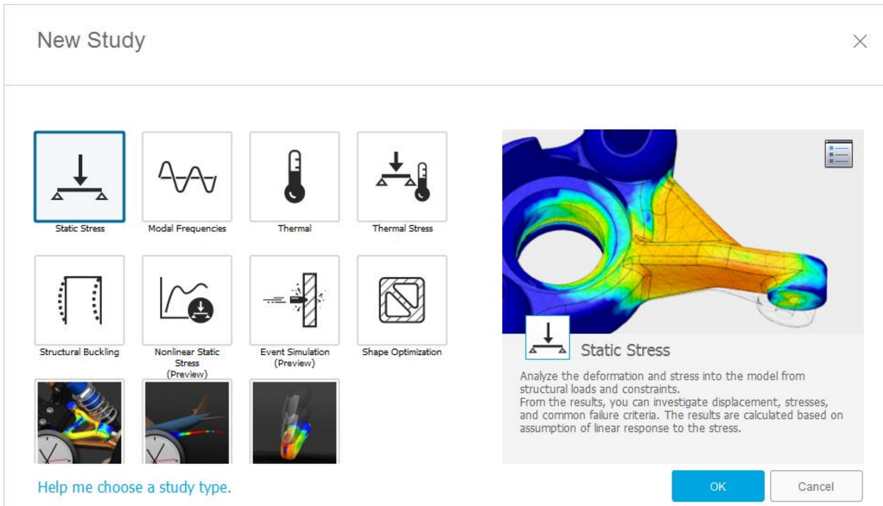


Figure 24 - New Static Stress study in Fusion 360

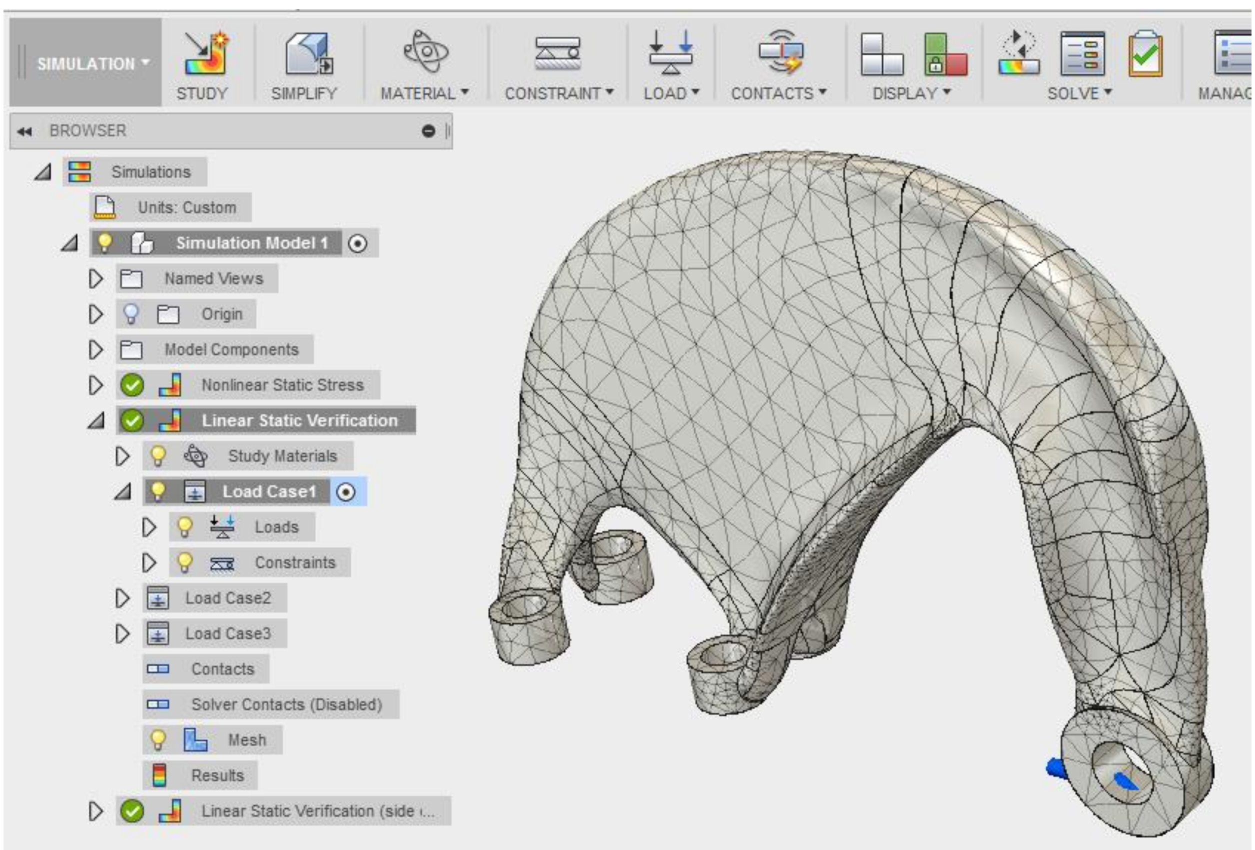


Figure 25 - Model prepared for simulation with three load cases

Material Definition

For best correlation, the exact AGD material properties should be reproduced in Fusion 360. This is done by creating a new material in the material browser and entering the physical properties in the 'Basic Properties' tab as shown below.

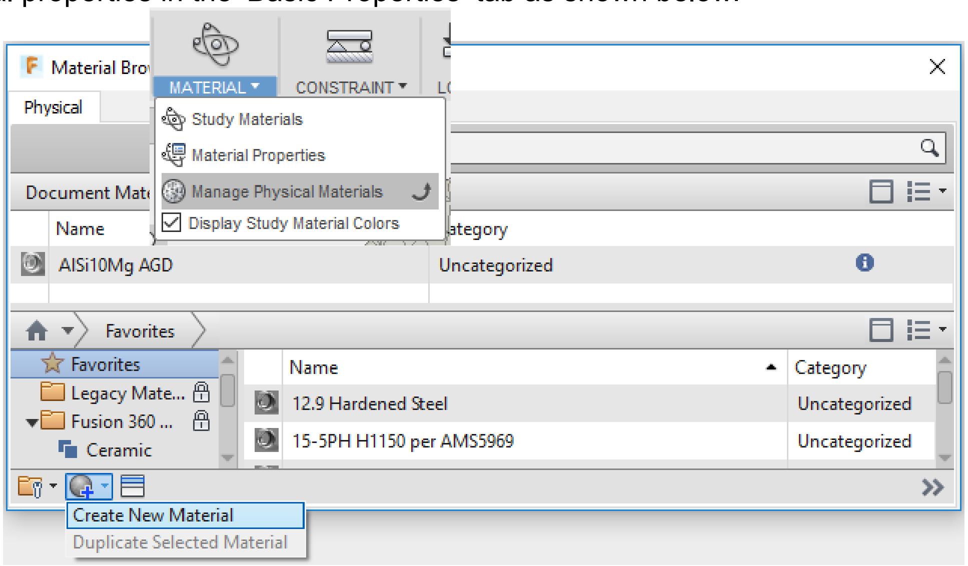


Figure 26 - Creating a new material in the material browser

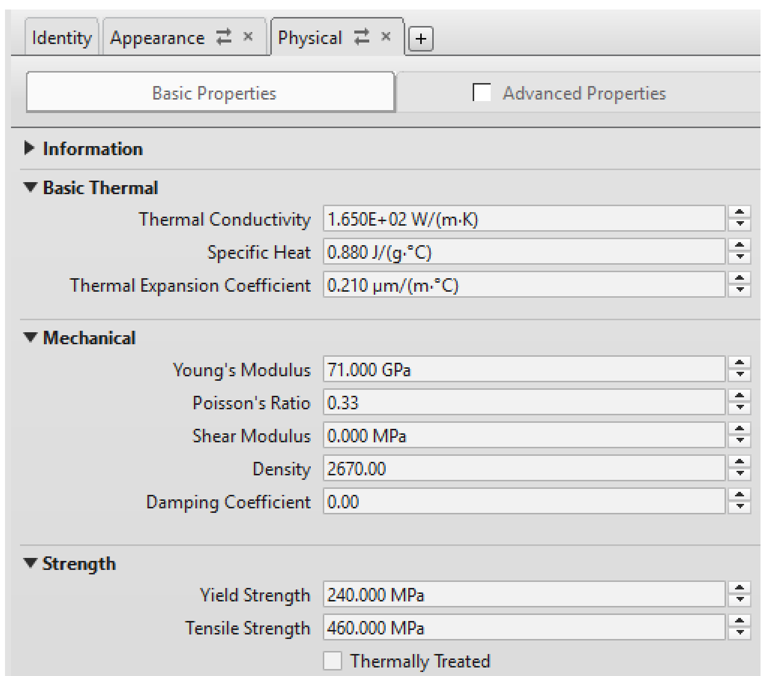


Figure 27 - Defining the physical properties in the material browser

Linear Static Solution Results

Load Case 1 (0°)

Load case 1 showed a maximum stress of 333.3 MPa at the location indicated below. This is the expected location for maximum stress and is consistent with the results from AGD.

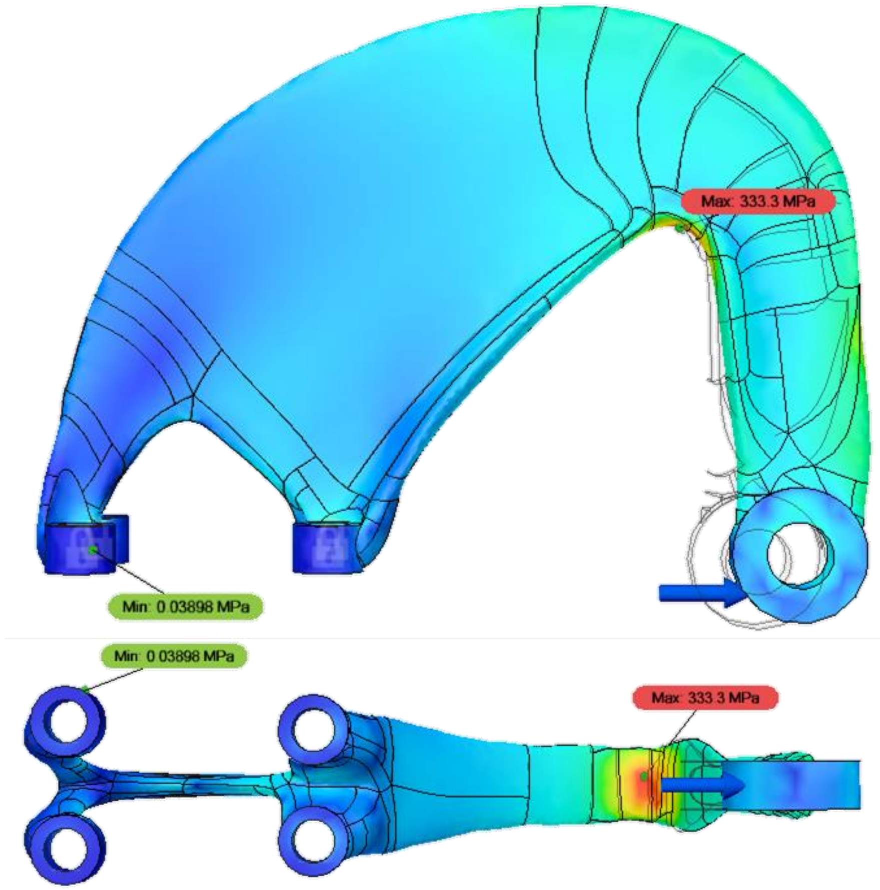


Figure 28 - Load Case 1 (0°) von Mises stress distribution

Figure 29 - Load Case 1 (0°) volume with SF≤1 [von Mises stress vs. yield strength]

While the maximum stress exceeds the material yield strength of 240MPa, it is important to consider the extent of the affected volume. By using a safety factor view and isolating all regions with safety factor less than or equal to 1, it can be seen that the affected volume is very local.

Load Case 2 (45°)

Load case 2 showed similar behavior to load case 1 with some higher stresses near the attaching bolts.

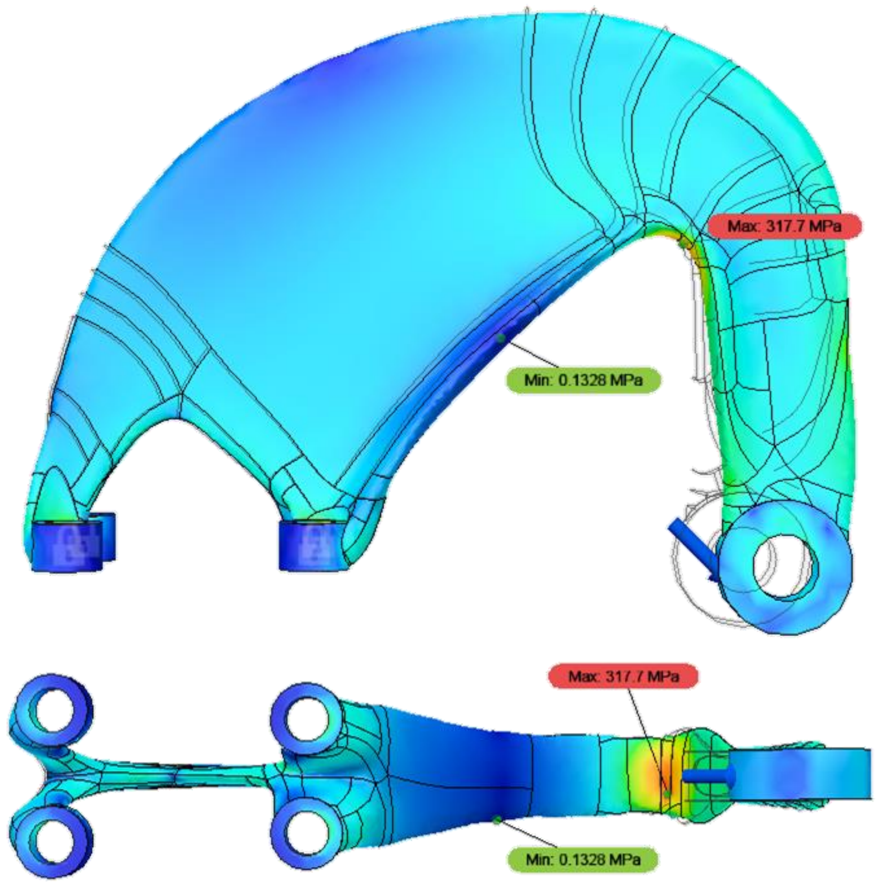


Figure 30 - Load Case 2 (45°) von Mises stress distribution

Similar to Load Case 1, it can be seen that the affected volume with safety factor less than or equal to 1 is very local.

Figure 31 - Load Case 2 (45°) volume with SF≤1 [von Mises stress vs. yield strength]

Load Case 3 (90°)

Load case 3 has a different critical area to the previous cases. The maximum stress occurs adjacent to the bolted interface. Again, there are local exceedances of the material yield strength.

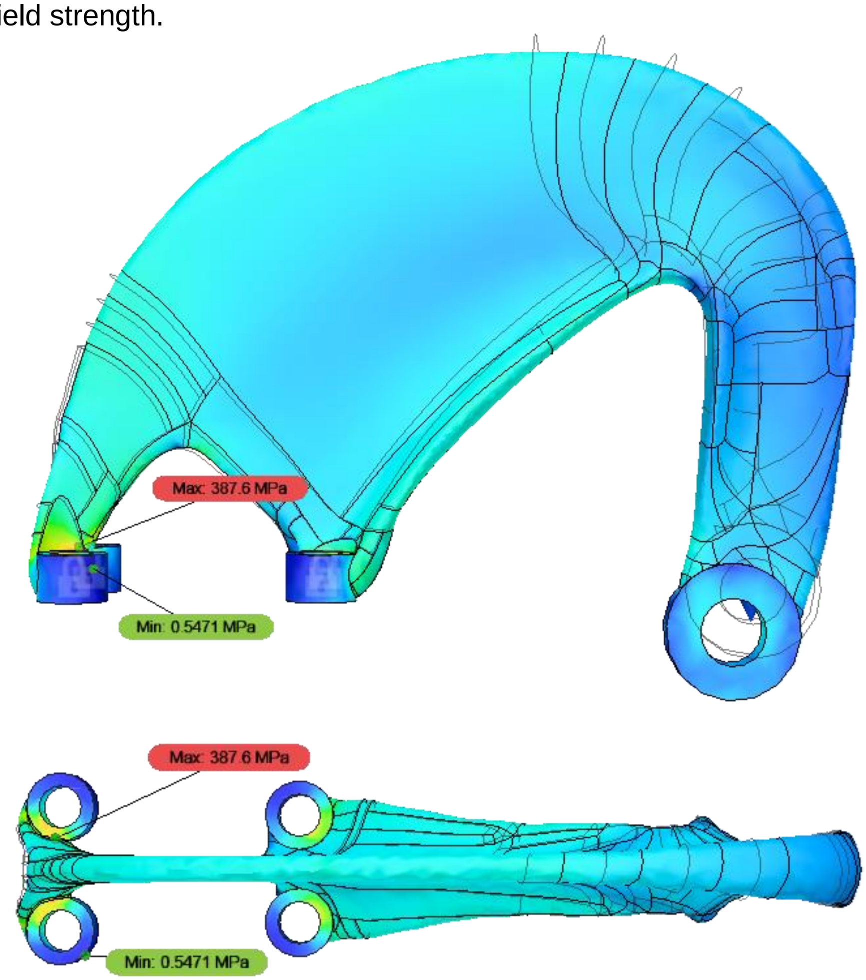


Figure 32 - Load Case 3 (90°) von Mises stress distribution

As shown below, the exceedances are very local, these occur for a number of reasons:

- There are sharp edges in the geometry and potentially poorly shaped elements
- Some material has been removed from the AGD design to allow access to the fasteners
- There are local nodal constraints which will give rise to high local stresses

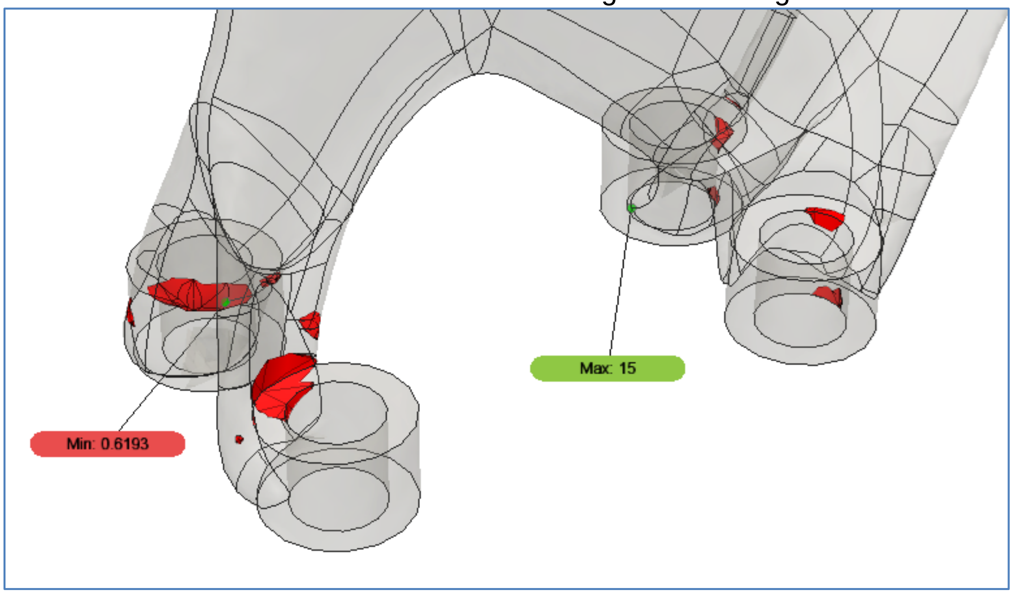


Figure 33 - Load Case 3 (90°) volume with SF≤1 [von Mises stress vs. yield strength]

Concluding Remarks

While there are few areas where the yield stress has been exceeded (and hence the AGD stress constraint violated), the affected volumes are very local and do not necessarily indicate gross yield. Should this bracket be applied in service, the following recommendations apply:

- Consider increasing the size of the attaching interfaces to ensure that enough material can grow from them
- Should absolutely zero yield be desired, the user should consider using a higher safety factor in AGD
- Ensure that the material properties being verified against accurately represent the as-built material

Generative Design and AM vs. Traditional Design and Subtractive Manufacturing

Specimens

The original bracket was machined from Aluminum 2014-T651 at the Autodesk Advanced Manufacturing Facility (AMF) in Birmingham, UK. As mentioned earlier, the spherical bearing was replaced by a simple 10mm diameter hole. Three specimens were manufactured.

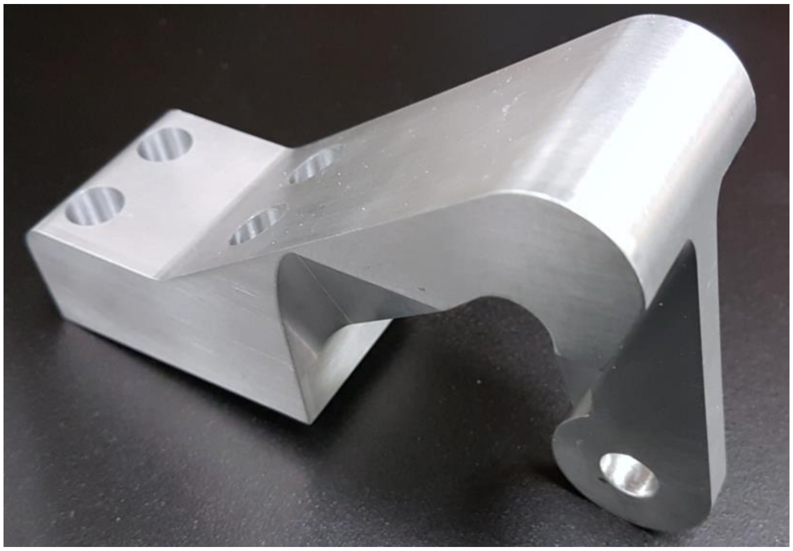


Figure 34 - Original (machined) bracket from GrabCAD challenge

The additive manufactured brackets were produced externally with Aluminum AlSi10Mg powder on an EOS M280 DMLS. A total of four unrestricted brackets were built, this included 3 for testing and 1 for display purposes. The +Y bracket was built for display purposes only.

Figure 35 - Generative Design AM Brackets (display models with logo)

Material Characterization

When performing mechanical tests, it is important to ensure that the assumed material properties match those that the part was built from. For this study two materials were used: (1) Aluminum Alloy Plate 2014-T651, and (2) Aluminum Powder AlSi10Mg. The supply materials were characterized by testing dog bone specimens and generating the applicable stress-strain curve.

Aluminum 2014-T651

Three dog bone specimens were cut from the stock material that was used to make the baseline GrabCAD bracket. The material curve and non-linear simulation input data is shown in the image below.

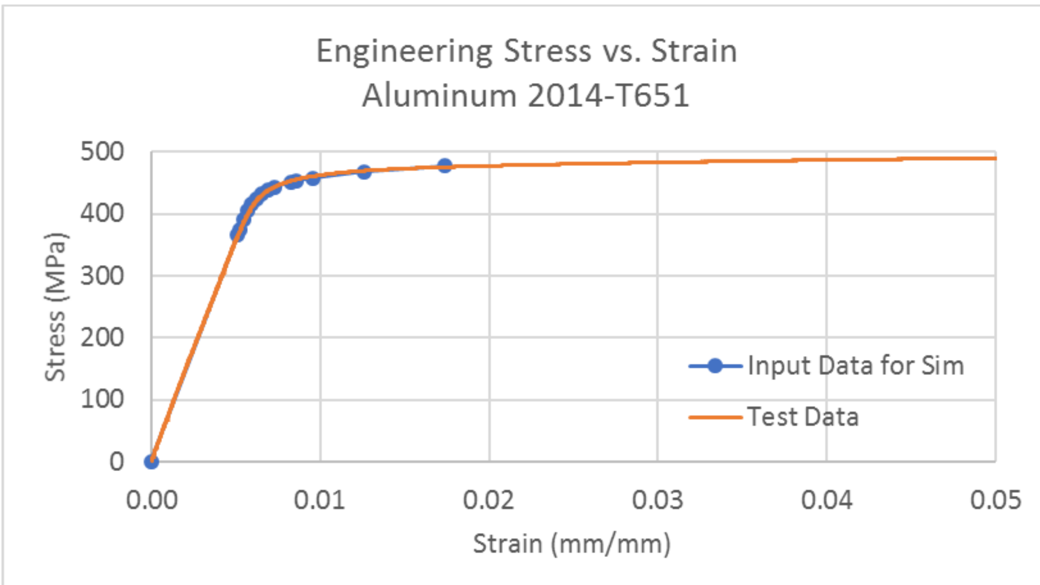


Figure 36 - Stress-strain curve for Aluminum 2014-T651 as tested

The properties calculated from the test data are:

- Elastic modulus: E = 72000 MPa
- Yield strength: $f_{0.2}$ = 451 MPa
- Ultimate strength: f_u = 477 MPa

Aluminum Powder AlSi10Mg

Three dog bone specimens were printed in the Z orientation and three in the XY orientation in the same build as the generative design additive manufactured brackets. The material curve and non-linear simulation input data is shown in the image below.

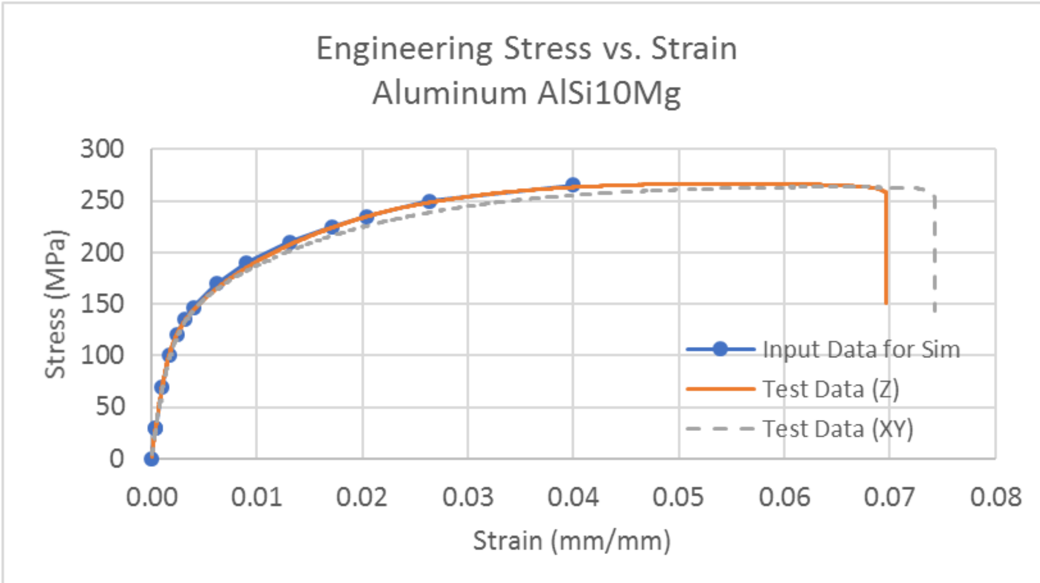


Figure 37 - Stress-strain curve for Aluminum powder AlSi10Mg as tested

The properties calculated from the test data (Z) are:

- Elastic modulus: E = 75000 MPa
- Yield strength: $f_{0.2}$ = 147 MPa
- Ultimate strength: f_u = 265 MPa

Test Apparatus

To enable testing of all three load cases (at 0°, 45° and 90°), a bespoke jig was designed using Fusion 360. This consists of a base frame, an interface fixture, and a load arm. The base frame attaches directly to the Universal Testing Machine (UTM) frame and the load arm connects to the UTM cross head. The load arm also has attachment points for a linear variable displacement transducer (LVDT) to allow measuring of local displacements. The interface fixture is designed to be positioned at all three angles without any movement of the load arm.

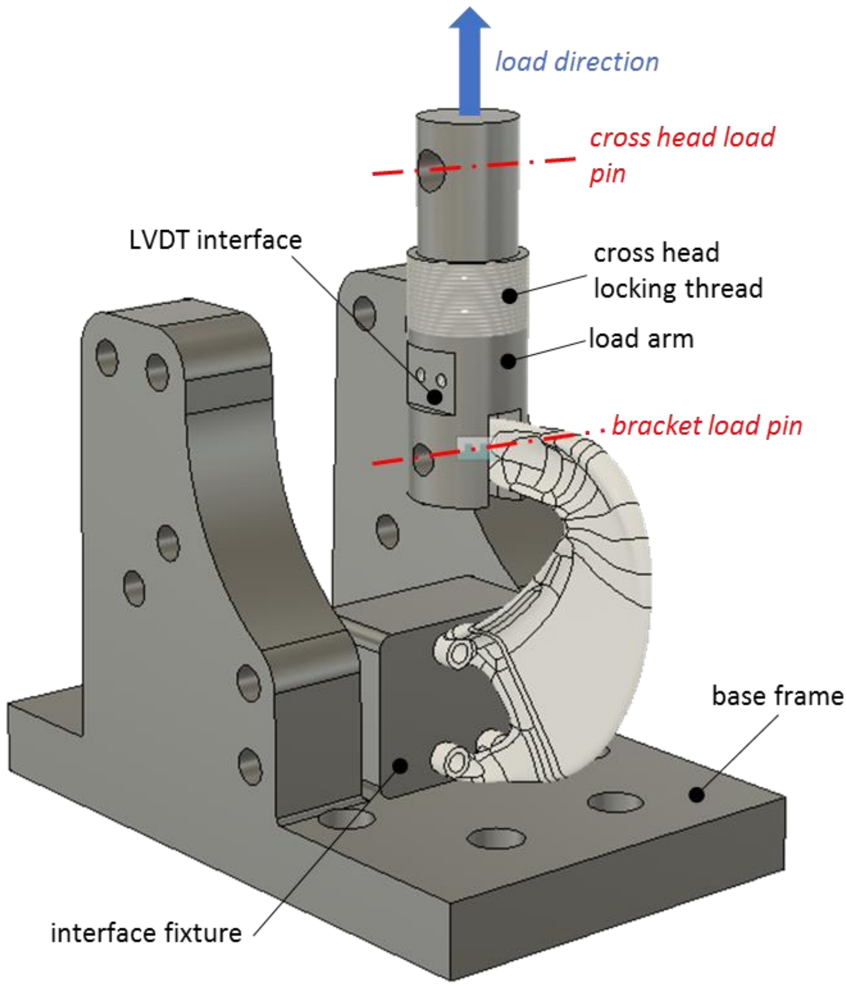


Figure 38 - Test apparatus as designed in Fusion 360

The jig was manufactured from medium steel and sized to minimize any deflection of its own components during test. It was also produced in the Autodesk AMF.

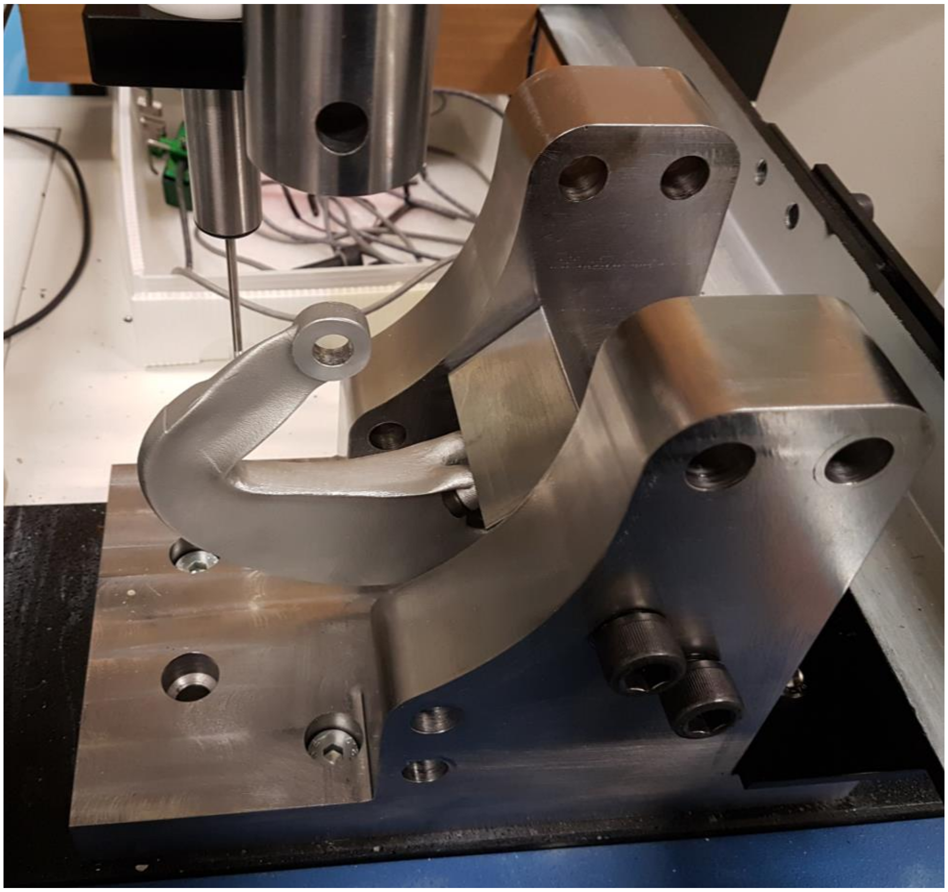


Figure 39 - As-manufactured test apparatus

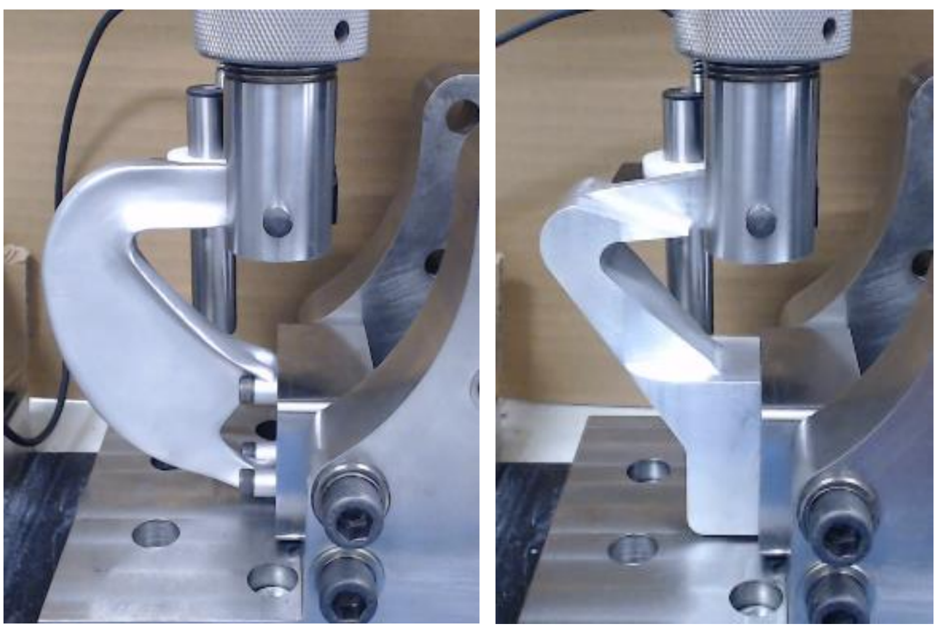
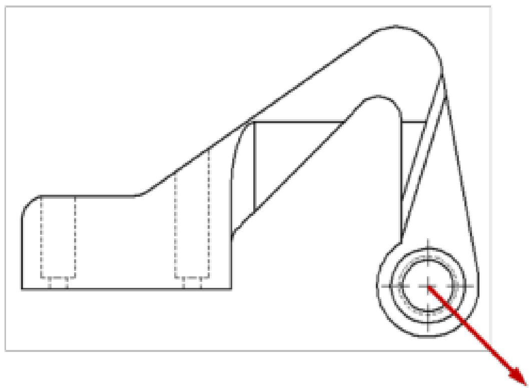


Figure 40 - Loaded generative design bracket (left) and loaded original bracket (right)

Results

Each bracket was tested in a specific sequence to maximise the data obtained. The intent was to obtain linear deformation data (i.e. no permanent deformation) for load cases 2 and 3 (45° and 90°), and then load at 0° (case 1) to failure. For this reason, the results for cases 2 and 3 are discussed before those for load case 1.

Load Case 2 (45°)



Each curve in the load-displacement graph below represents an average of three different specimens. The machined bracket has a steeper slope indicating that it is about 10% stiffer than the generative design bracket.

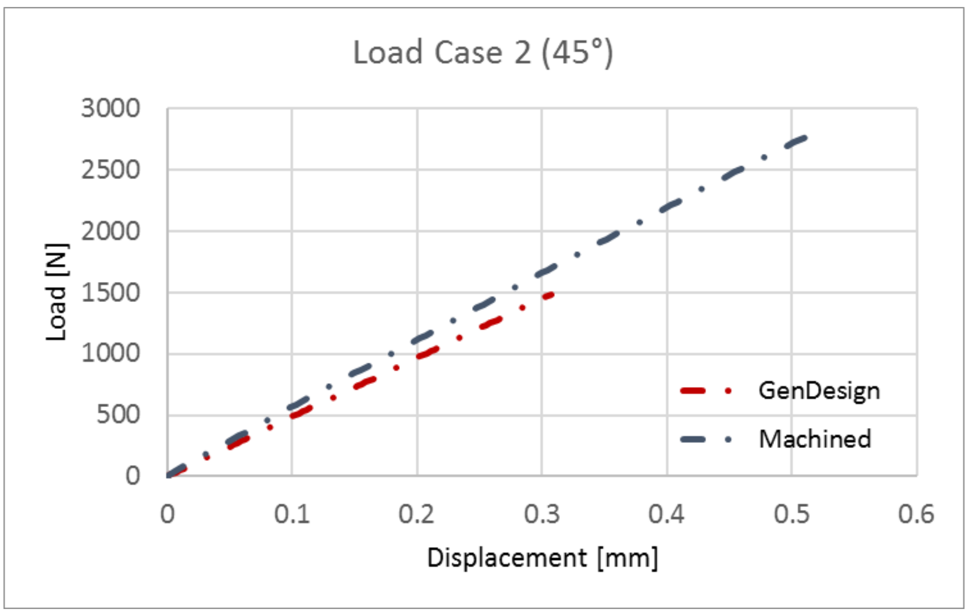
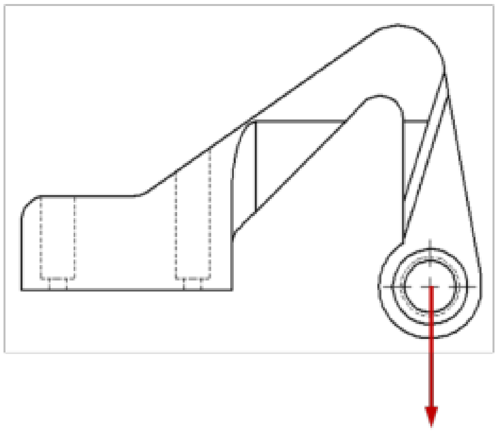


Figure 41 - Load-displacement test averages for load case 2

Load Case 3 (90°)



Like the previous graph, each curve in the load-displacement graph below represents an average of three different specimens. In the linear region, the slopes are almost identical meaning that the two brackets have about the same stiffness. It is noted though, that the onset of plastic deformation has occurred earlier for the generative design bracket (see the diversion of the red curve.

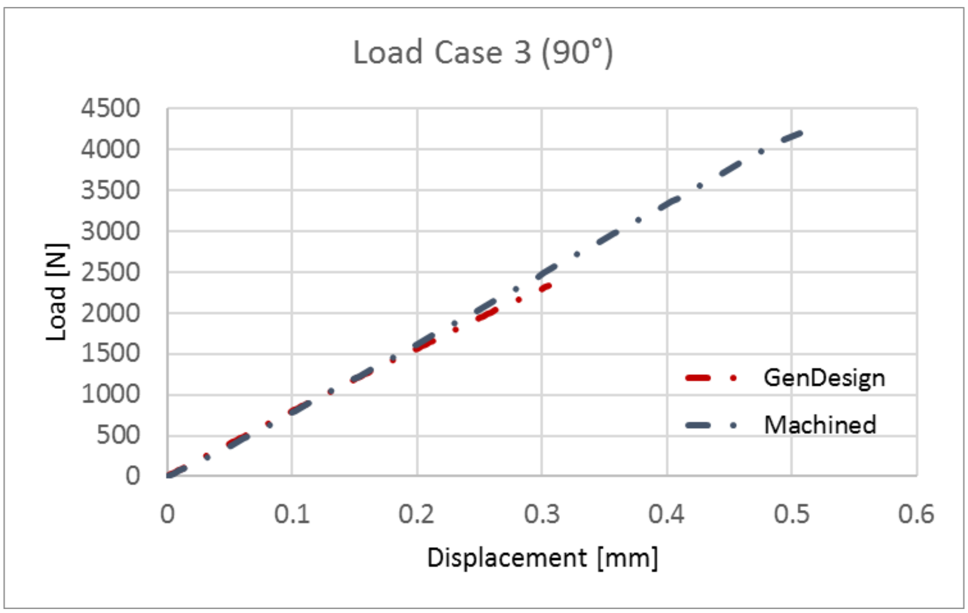
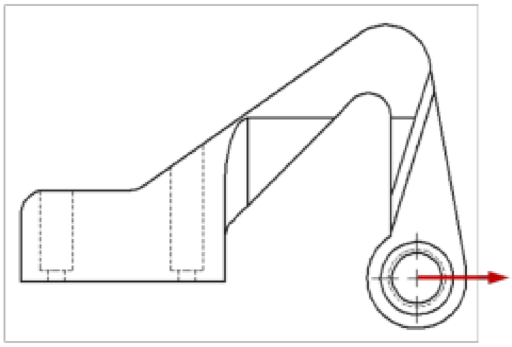


Figure 42 - Load-displacement test averages for load case 3

Load Case 1 (0°)



Original Machined Bracket

The original machined bracket behaved linearly for much of the case 1 load. It had a poor elongation to failure largely due to a crack initiated at a sharp feature (see image after graph below).

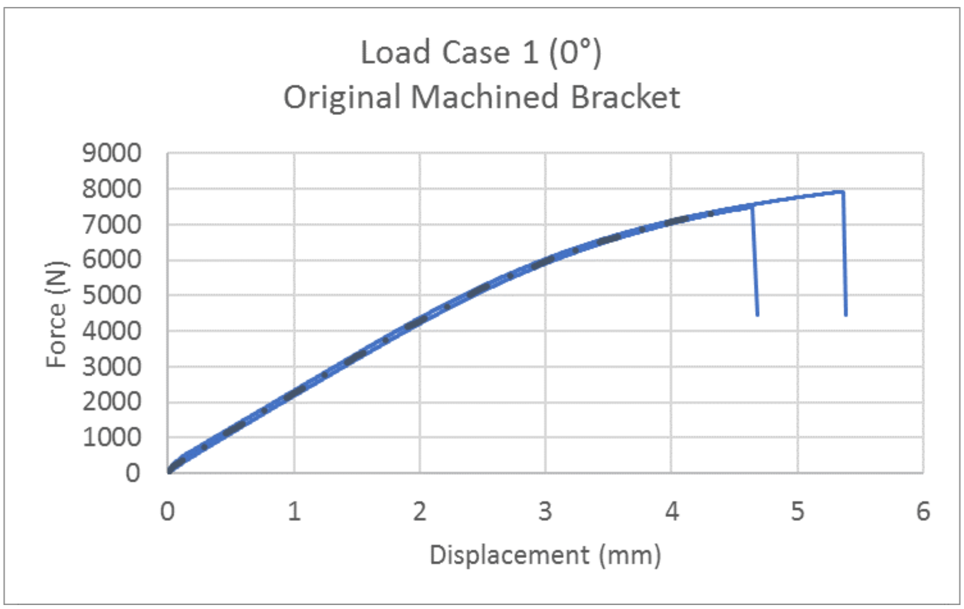


Figure 43 - Load-displacement curves for machined bracket, case 1

APPROXIMATE LOAD AT FRACTURE: **7700 N**

NORMALIZED TO ULTIMATE STRENGTH 500 MPa:

7700 x 500 / 451 = **8537 N**

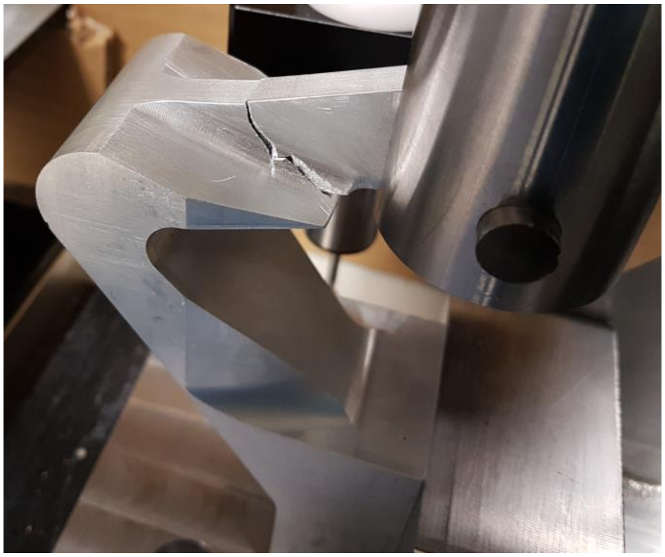


Figure 44 - Failure location of machined bracket

Generative Design Bracket

The generative design bracket showed good elongation, especially given that the AM material is generally less ductile. The onset of plasticity occurred early as expected, and the failure location was at the underside of the kink, also as expected.

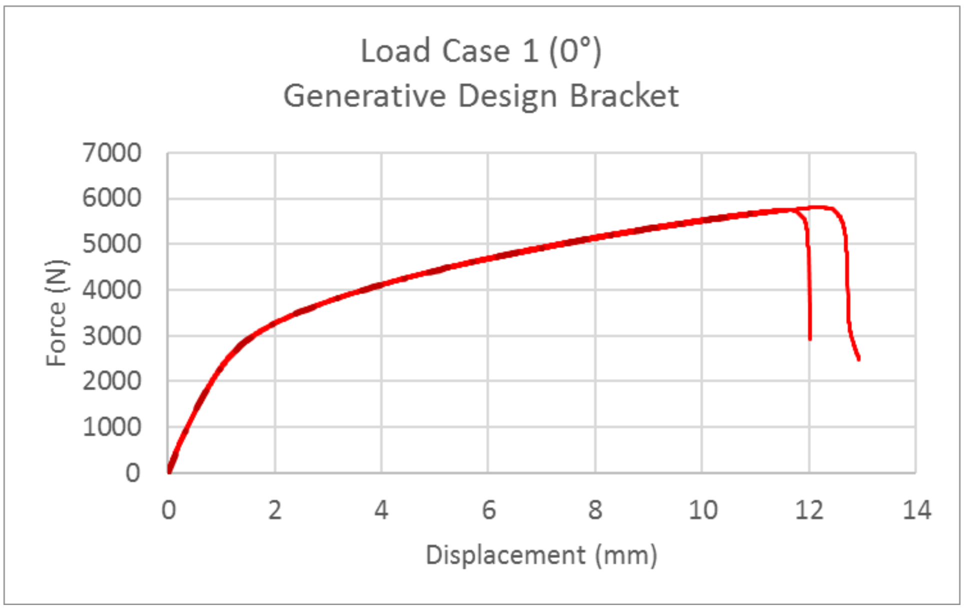


Figure 45 - Load-displacement curves for generative design bracket, case 1

APPROXIMATE LOAD AT FRACTURE: **5600 N**

NORMALIZED TO ULTIMATE STRENGTH 500 MPa:

5600 x 500 / 265 = **10566 N**

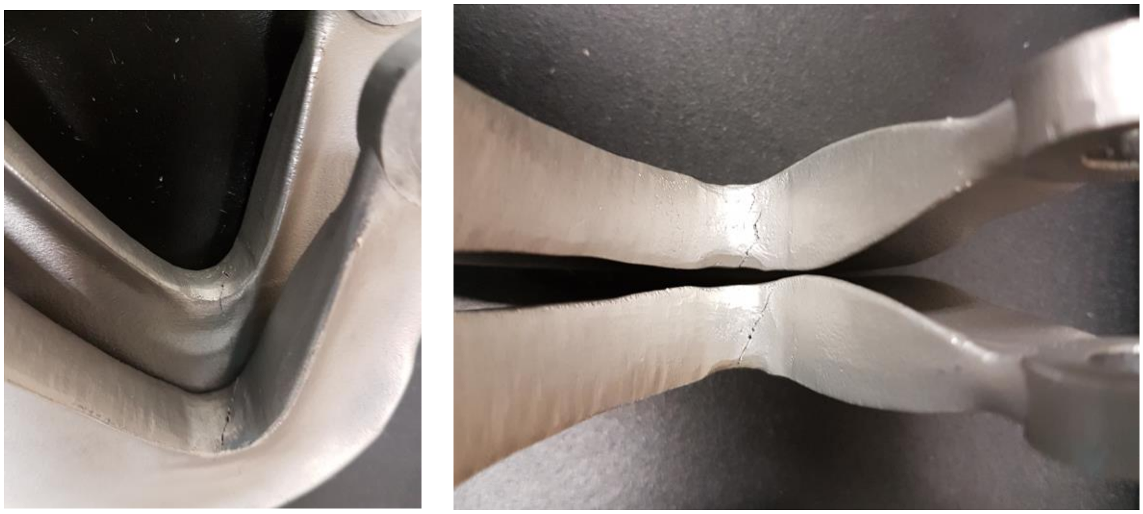
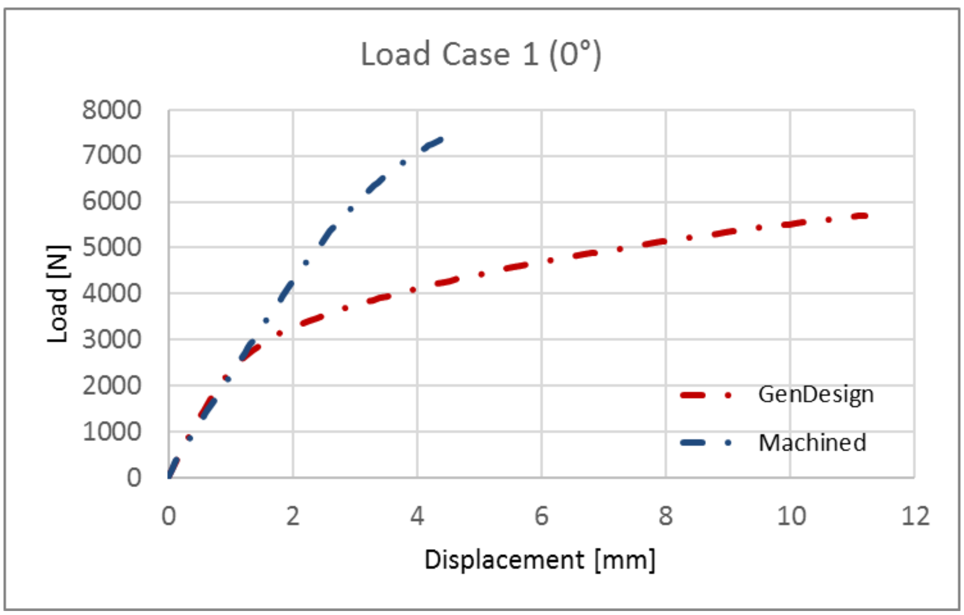


Figure 46 - Failure location of generative design bracket

Original Machined vs. Generative Design Additive Manufactured



Considering that the generative design bracket has a greatly reduced yield strength than that of the machined bracket (147 MPa vs. 451 MPa), it's performance very good. The following observations apply:

- The measured masses were 83.7g (generative) and 302.8g (machined) showing that the generative design bracket is only 28% of the mass of the machined bracket.
- In cases 1 and 3, the linear stiffness was equal between the brackets and in case 2, the generative design bracket had a reduction of only 10%.
- The (case 1) plastic deformation of the generative design bracket lasted much longer than the machined bracket. This indicates efficient placement of material.

- When considering the ultimate failure loads normalized to the material ultimate strength, the generative design bracket is 24% stronger.

Correlation with Simulation

Setting up a Non-linear Static Stress Simulation in Fusion 360

Non-linear Static Stress (NLSS) study type can be created with the new study dialogue. Note that this currently remains a technical preview.

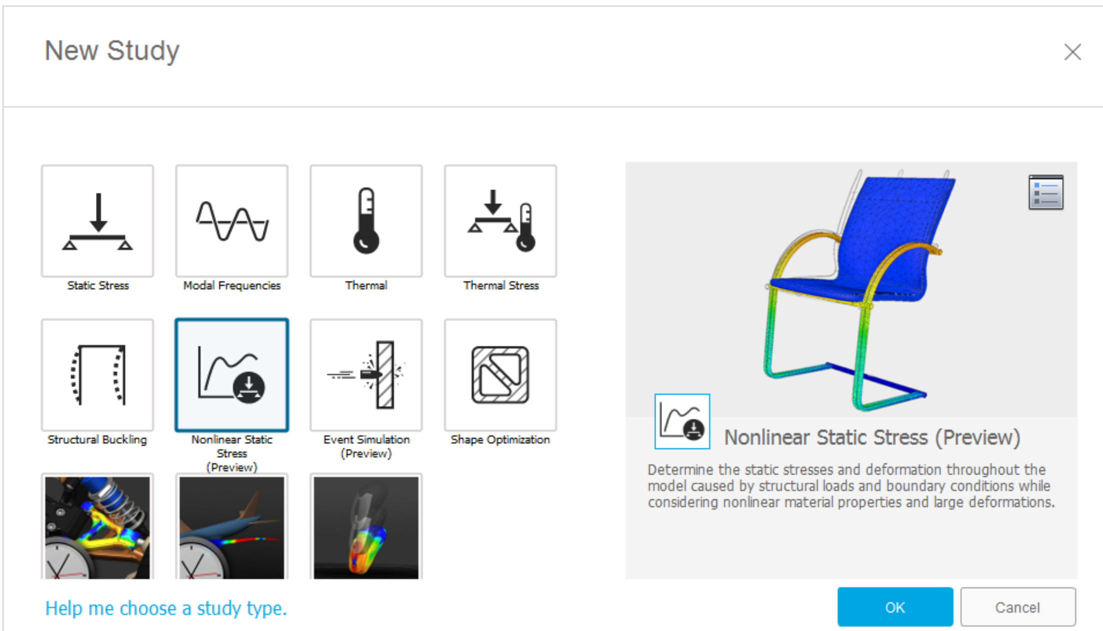


Figure 47 - Selecting NLSS study type in Fusion 360

Set up of the loads and constraints does not differ from the static stress study previously described. However, the number of load steps needs to be specified; a higher number of load steps generally means higher accuracy however it will take longer and generate much more data. There are also some important differences to note with the material definition that are explained below.

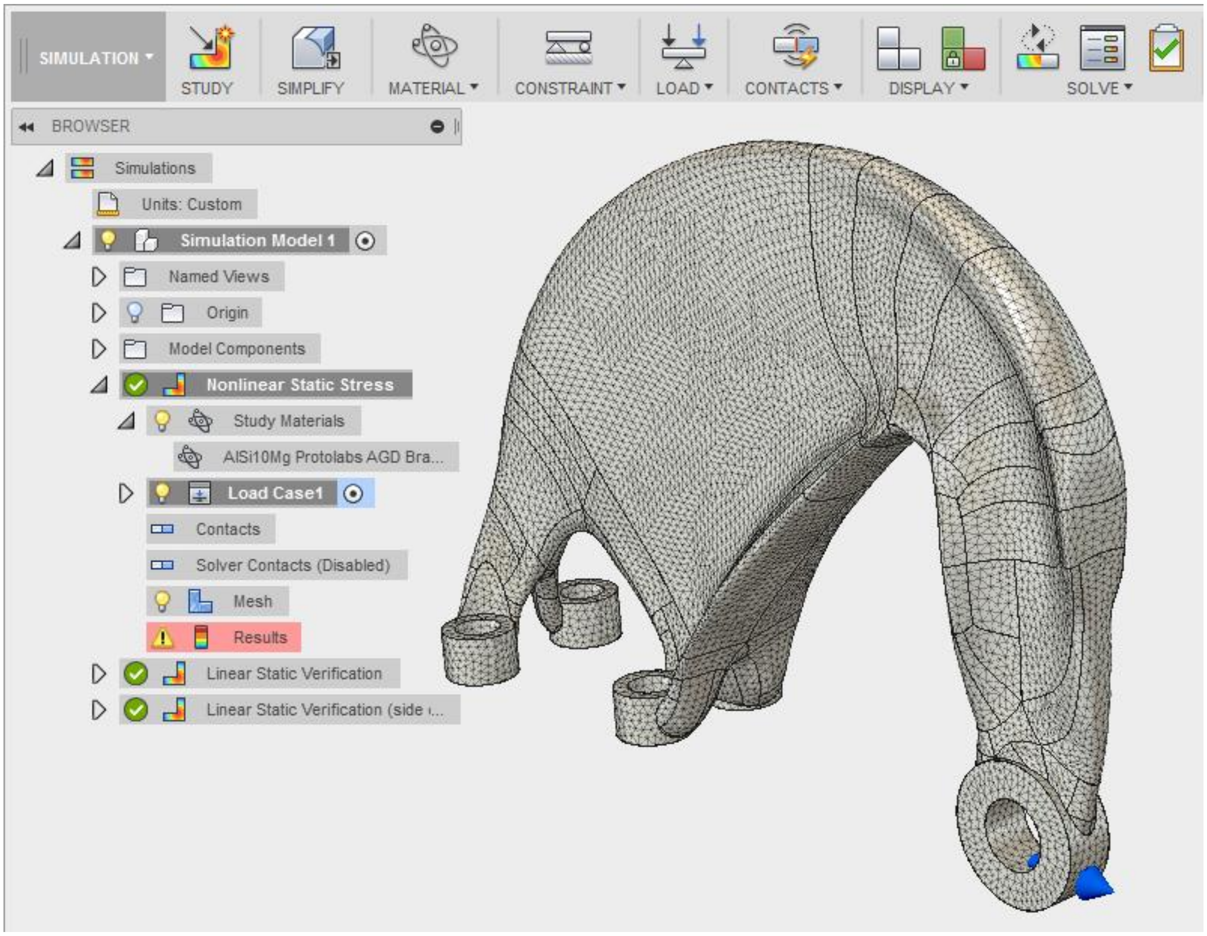


Figure 48 - Meshed model with loads and constraints applied

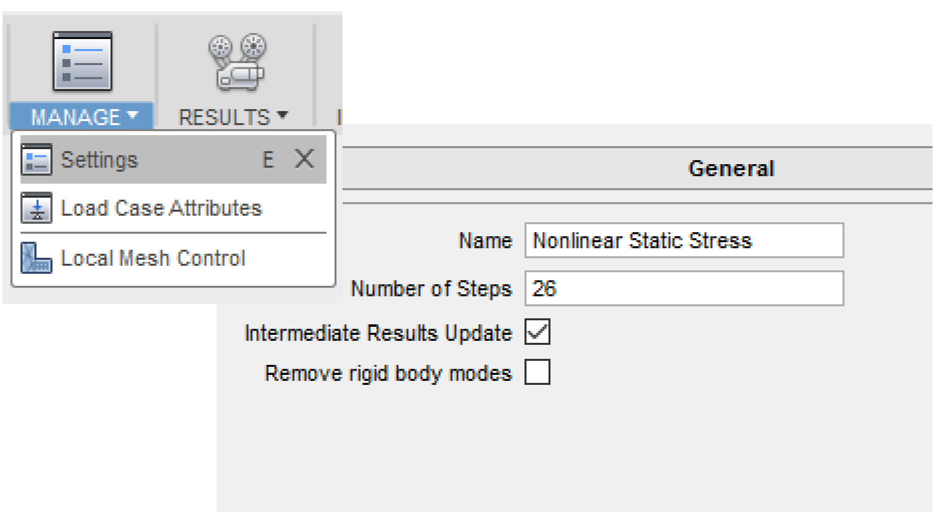


Figure 49 - Setting the number of steps (load increments)

Material Definition

To simulate an elastic-plastic material, the stress-strain curve data needs to be entered in addition to the basic elastic properties. For the tested AlSi10Mg material, the following data points were chosen to describe the stress-strain curve.

Table 1 - Data points describing the stress-strain curve

Strain	*Stress (MPa)*	
0.00000	*0.0*	
0.00040	*30.1*	Initial yield stress
0.00100	70.0	
0.00167	100.0	
0.00239	120.0	
0.00315	135.0	
0.00396	147.0	
0.00617	170.0	
0.00904	**190.0**	0.2% yield strength
0.01312	210.0	
0.01717	225.0	
0.02044	235.0	
0.02634	250.0	
0.04000	**265.0**	Ultimate strength

It is important to note that the second data point indicates the initial stress whereby the stress begins to vary non-proportionally with the strain. The slope of the line from (0,0) to (0.0040, 30.1) must be equal to the elastic modulus which in this case is 75 GPa. These data are entered into the 'Advanced Properties' tab when creating a new material in the material manager.

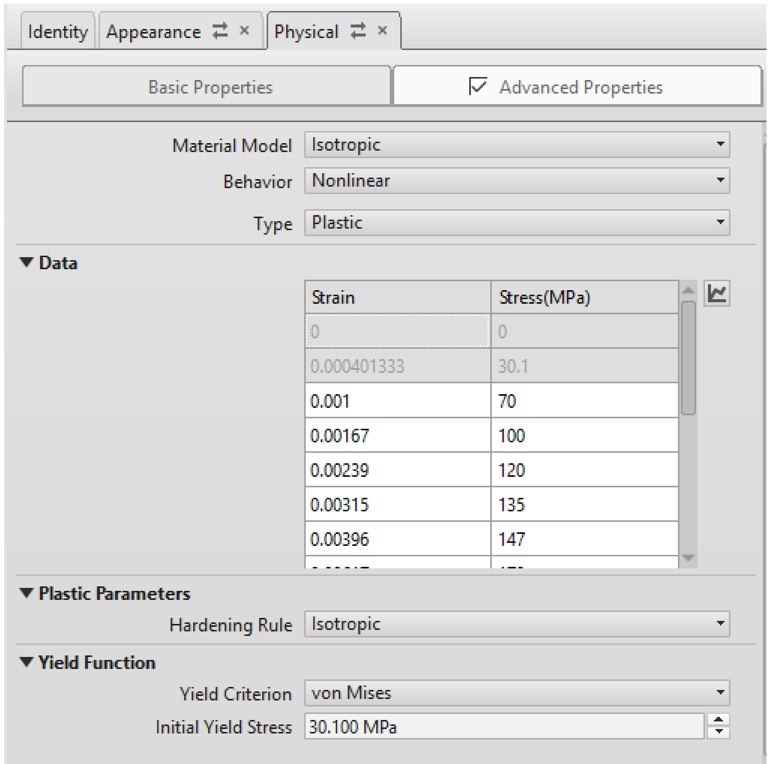


Figure 50 - Entering non-linear stress-strain data

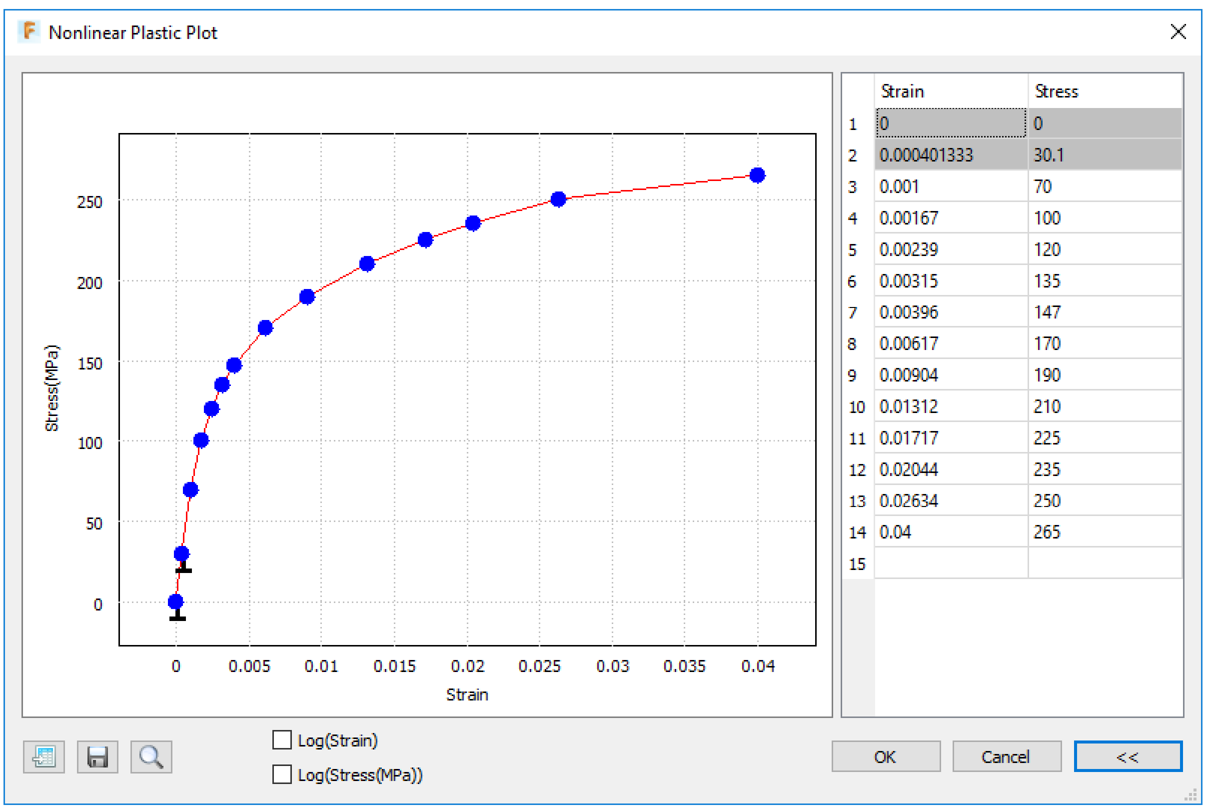


Figure 51 - Plot of entered material properties

NLSS Results and Correlation

The following results show very good correlation of the measured load-displacement at the load-pin location. These were measured by taking a probe on the surface of the bracket load lug shown in the subsequent image.

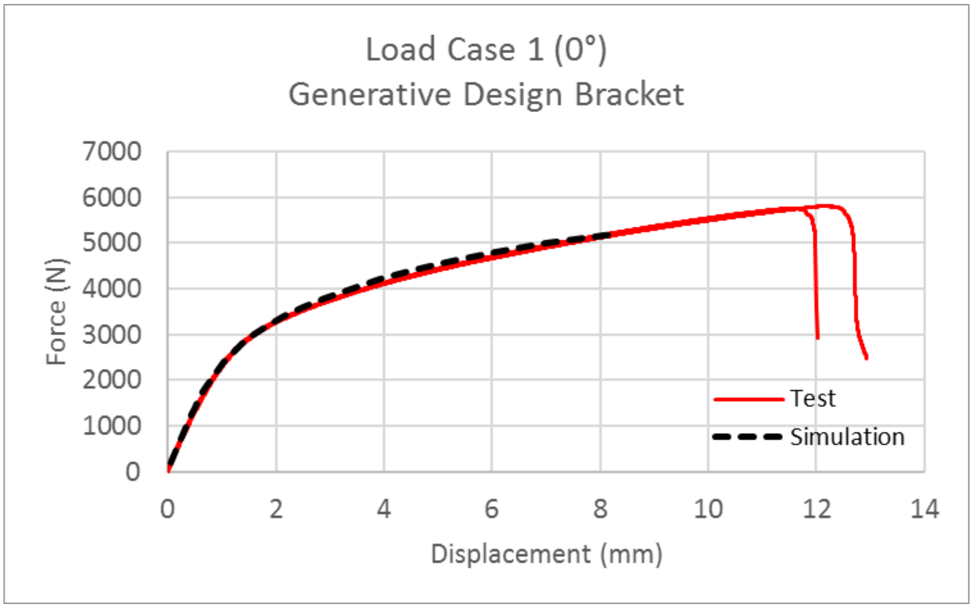


Figure 52 - Correlation of force-displacement simulation at the load lug

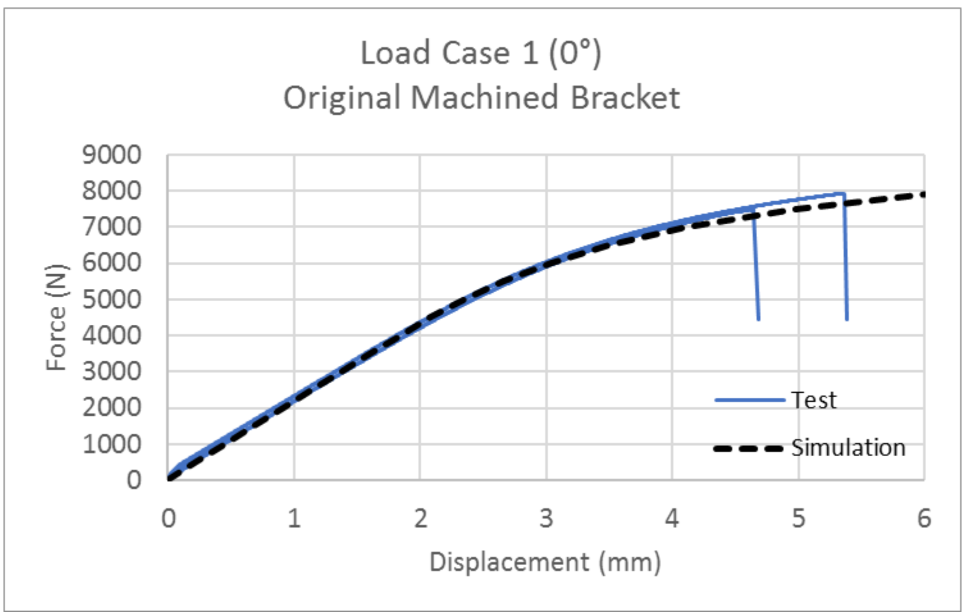


Figure 53 - Correlation of force-displacement simulation at the load lug

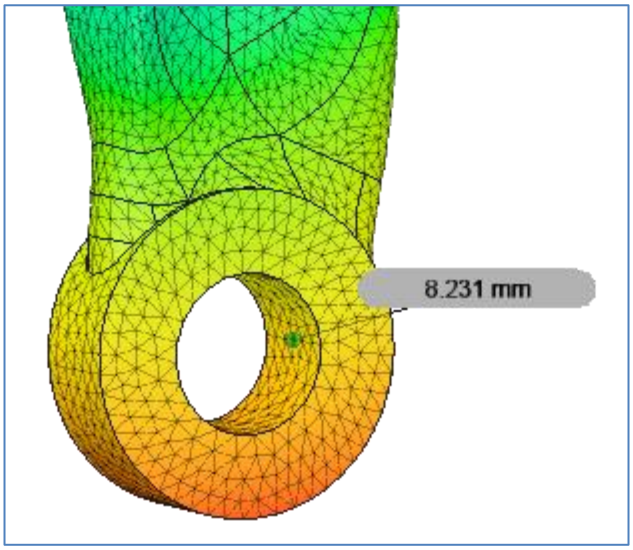


Figure 54 - Using a surface probe in Fusion 360

Performance Comparison

The good correlation of force and displacement of both brackets gives confidence in the derived stress results from the NLSS simulation. These are thus used to draw some comparisons between the two brackets. There are a couple of main load points to consider:
(a) the load at which onset of yield (or permanent deformation) occurs; this is determined by checking when the yield strength $F_{0.2}$ is first exceeded, and (b) the load at which final fracture initiates, i.e. ultimate strength F_u is first exceeded.

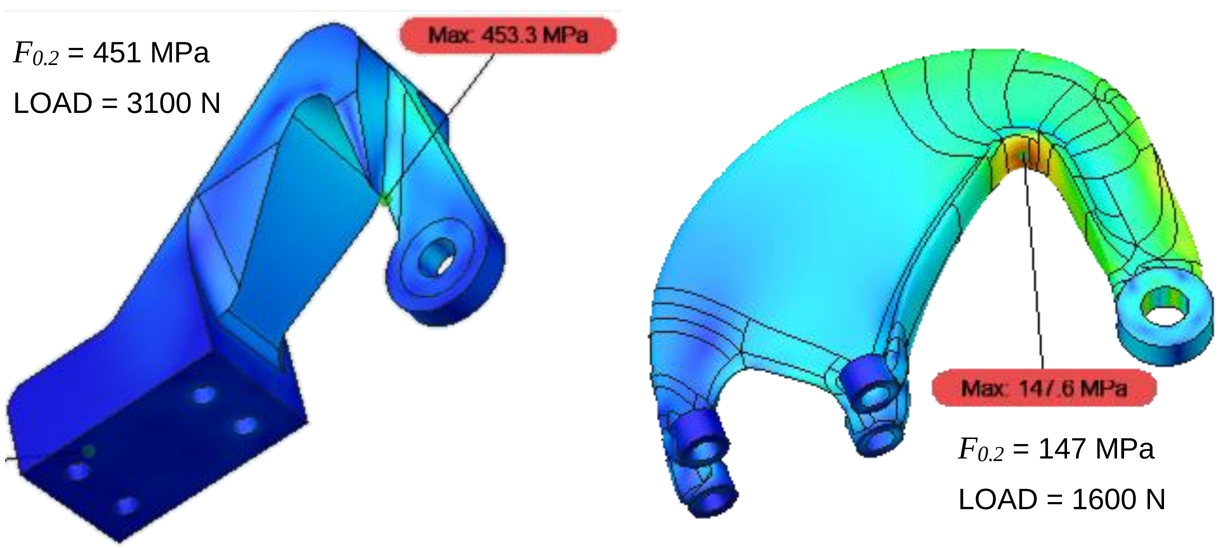


Figure 55 - Point at which yield is first exceeded

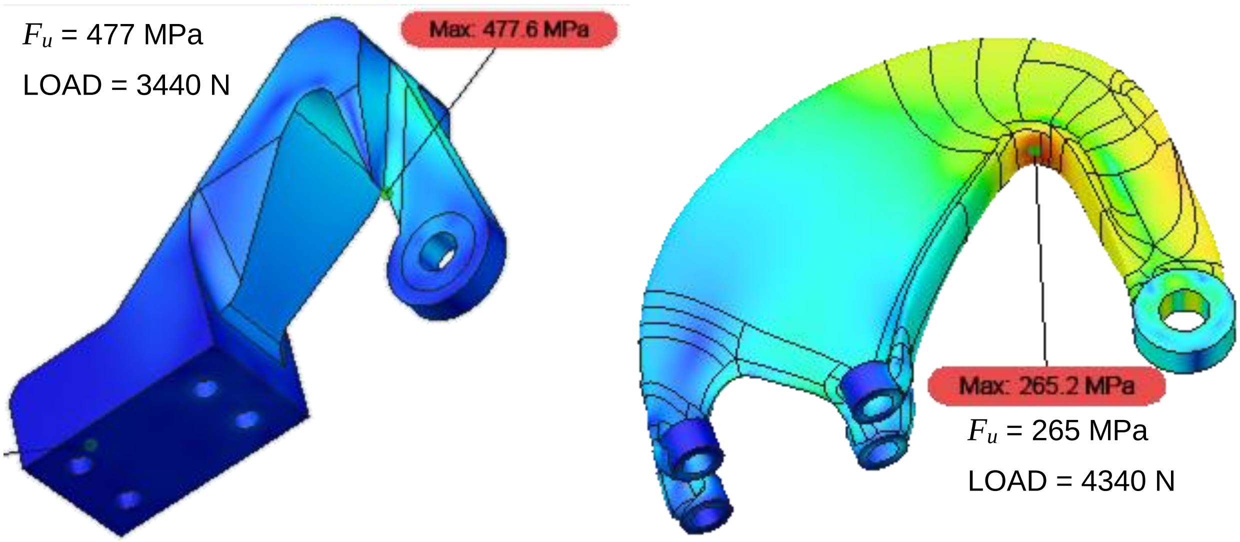


Figure 56 - Point at which ultimate strength is first exceeded

As indicated by the test, the original bracket has a clear failure that initiates at the critical location shown above. While the material is stronger, failure of this bracket starts long before the generative design bracket. This is very important when considering fatigue performance.

Shape efficiency

In order to remove the material differences from the comparison, the original bracket was also simulated with the AlSi10Mg material. This approach gives a good indication of how much better the overall shape of the generative design part is. It also helps to quantify the performance improvement should a like-for-like material be used.

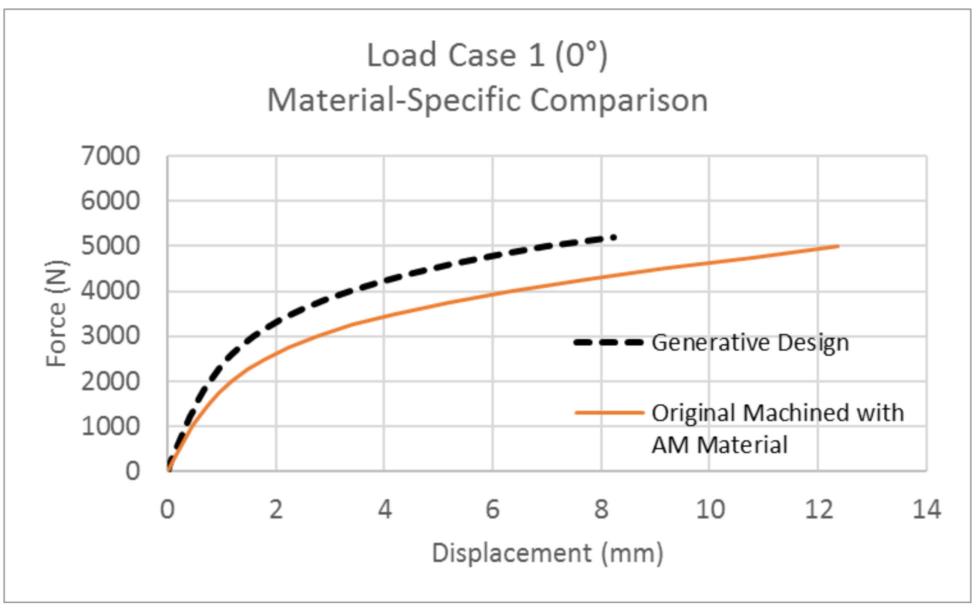


Figure 57 - Simulated force-displacement at the load lug for both brackets with the same material

In this simulation, the original bracket begins to yield at 1175N as compared to the generative design bracket at 1600N. As a measure of shape efficiency, the generative design bracket can be said to be 36% better.

Conclusions

Autodesk Generative Design provides the capability to generate multiple designs that can all be manufactured into a real product. Just one design of the many potential options was investigated in this handout. When used together with verification tools like Nastran non-linear static solver in Fusion 360, the engineer can very accurately simulate the real mechanical behavior of these designs.

While the original GrabCAD design used a far stronger material, the generative design bracket showed superior performance in many ways. Firstly, it’s shape efficiency is superior and allows stresses to redistribute thus enhancing ductility in the plastic range. A 36% improvement of the load at the onset of plastic deformation was realized. Next, there was a 24% improvement in the normalized ultimate load. Finally, the as-built generative design bracket weighed a mere 28% of the original bracket.

If this part were to be used in service, the final additive manufactured part would use a higher quality aluminum powder and be heat treated as required. Higher quality aluminum powder could achieve up to 270 MPa yield and 460 MPa ultimate thus the generative design bracket would see even better performance.