



PRIFYSGOL CYMRU
Y Drindod Dewi Sant
UNIVERSITY OF WALES
Trinity Saint David
SWANSEA - ABERTAWE



Design for X Project

MANUFACTURING, DESIGN & TECHNOLOGY
(ACMA5002)

Joshua U'Dell
BOC A LINDE COMPANY
UNIVERSITY OF WALES TRINITY SAINT DAVID

Contents

Contents.....	1
1.0 Introduction.....	2
2.0 Task 1 - DfAM Case Study.....	3
2.1 Original Component	3
2.2 3D Model.....	3
2.3 Finite Element Analysis (FEA).....	4
2.4 Topology Optimisation.....	13
2.5 Design for Additive manufacturing	17
2.6 Final Design.....	18
3.0 Task 2 – High-volume production – Portable Wind Turbine Redesign.....	24
Appendix	26
Appendix 1	26
Appendix 2.....	27
Appendix 3.....	28
References	29

1.0 Introduction

This project will be discussing and utilising Design for Additive Manufacturing (DfAM) techniques focusing on two separate scenarios. The first section of this report will be taking an existing component used within BOC, an industrial gasses supplier and manufacturer based in the United Kingdom and Ireland and applying DfAM principles to optimise the design for Additive Manufacturing (AM) whilst ensuring the component is still mechanically suited for the intended use case.

The second section of this project will be looking at a predesigned assembly that has mainly been designed to be produced using conventional plastic 3D Printers. The design needs to be analysed and modified for mass production.

2.0 Task 1 - DfAM Case Study

2.1 Original Component

The component this task will be looking at is a valve wall bracket design by the BOC Drawing Office in 2016, the bracket has the BOC part number/ reference 19328882. The detailed drawing for this can be found in appendix 1 of this document.

As can be seen in the drawing the bracket is made from 2 Rolled Steel Angles (RSA) that are 80 x 80 x 6 mm. these 2 RSA's are continuously welded together. There are 2 Ø18mm holes on one of the angles, this allows the bracket to be bolted to the desired wall. On the other angle there are 2 Ø12 x 10mm slots for anchoring the valve to the bracket. The vendor drawing for the valve can be seen in appendix 2 of this document.

BOC Assembly drawing 028-91-P-D0006 can be found in appendix 3 and shows how the valve and bracket are designed to be fitted together. The bracket is designed to hold the valve in place during maintenance tasks and to help ensure the concentricity of the valve ports with the pipework.

The original component is made from RSA Steel to BS EN 10025 GRADE S 275 and then hot dipped galvanised to BS EN ISO 1461. This process adds a layer of zinc to all external faces of the component and protects the steel from corrosion due to rust. FABORY (2023) was a good use of information for this.

2.2 3D Model

The component has been modelled in 3D using SolidWorks 3D CAD software. This was done by the BOC Design Office back in 2016 and stored in the appropriate document repository. The original model file was designed using the weldments feature in SolidWorks 3D. This feature works by drawing the wire frame of the structure and then applying a structural member to each line in the sketch. Using this feature the Design Office was able generate the model in the drawings mentioned previously. (SolidWorks, 2023)

2.3 Finite Element Analysis (FEA)

The first stage of the DfAM process was to run an FEA study on the original component, the software used was SolidWorks 3D CAD with the SolidWorks Simulation add in activated. This add in allows the user to simulate a load onto the component and the software runs numerical calculations to obtain stress, strain and deflection results that can be interpreted by the user. The first step of an FEA static study is to check the model geometry and make sure the software understands how the bodies are connected. As this component was designed and modelled using the weldments feature the software sees each RSA as separate bodies and doesn't understand that they are welded together. So to account for this interaction the 2 bodies are joined together, this tells SolidWorks to treat them as one single body.

The next step is to assign a material to the whole component. This is done by selecting the "apply material" feature and then selecting the appropriate material from the pre-defined list. Galvanised steel was selected and the mechanical properties for this material choice can be seen in figure 1 below.

Material properties
Materials in the default library can not be edited. You must first copy the material to a custom library to edit it.

Model Type: **Linear Elastic Isotropic** ☐ Save model type in library

Units: **SI - N/m² (Pa)**

Category: **Steel**

Name: **Galvanized Steel**

Default failure criterion: **Max von Mises Stress**

Description:

Source:

Sustainability: **Defined**

Property	Value	Units
Elastic Modulus	2e+11	N/m ²
Poisson's Ratio	0.29	N/A
Shear Modulus		N/m ²
Mass Density	7870	kg/m ³
Tensile Strength	356900674.5	N/m ²
Compressive Strength		N/m ²
Yield Strength	203943242.6	N/m ²
Thermal Expansion Coefficient		/K
Thermal Conductivity		W/(m·K)
Specific Heat		J/(kg·K)
Material Damping Ratio		N/A

Figure 1 SolidWorks Galvanised Steel Properties

The next step of the FEA study is allocating the fixture points of the model, these are the faces and edges that can not move due to how the component is fixed/ secured. This part is secured using 2 wall bolts, to simulate this the faces on the inside of the holes are selected as the fixture points as shown in figure 2.

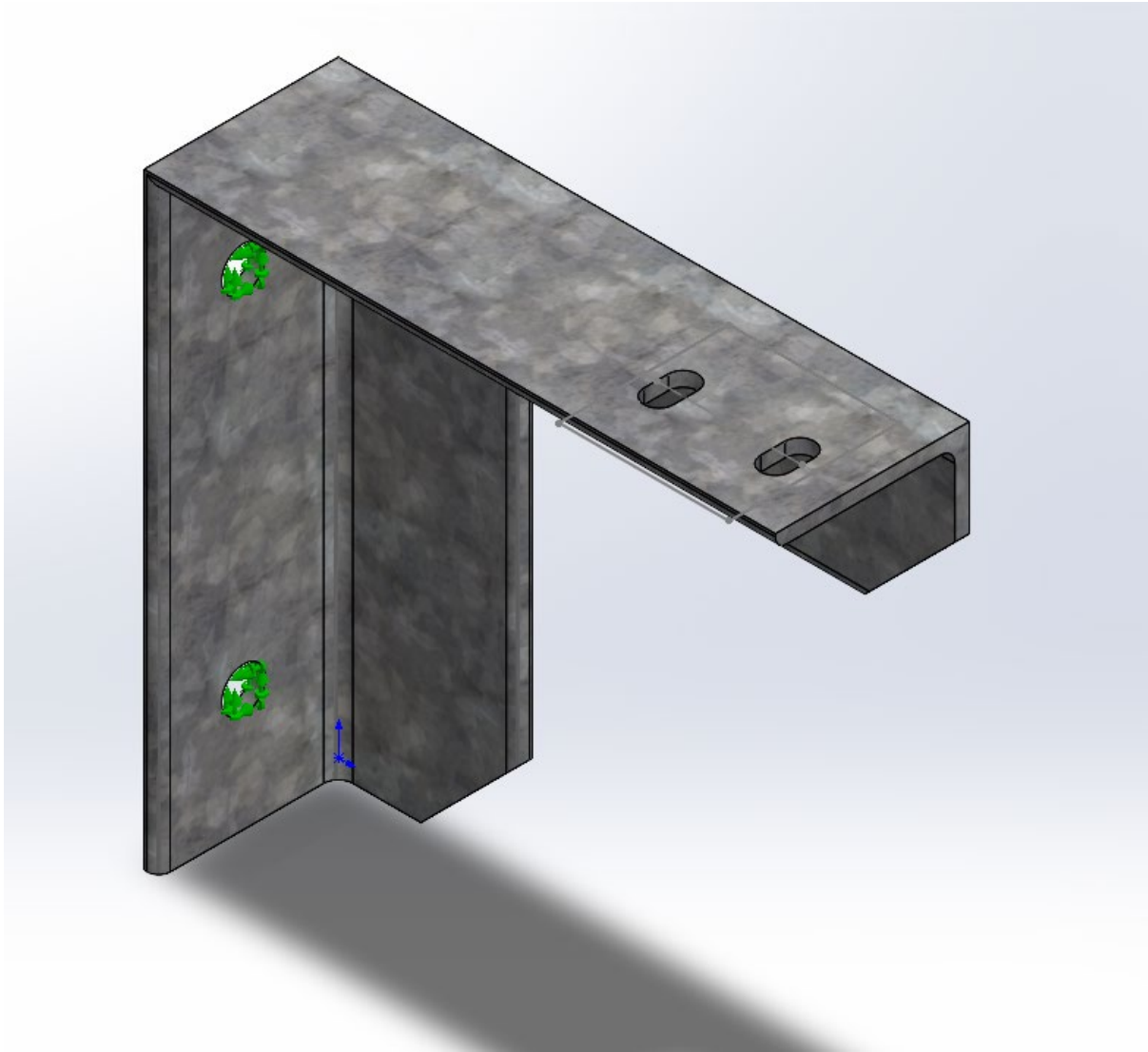


Figure 2 Static study fixture points

To simulate the force of the valve on the part a sketch is drawn on the top face at the size and location of the bottom face of the valve as shown in figure 3.

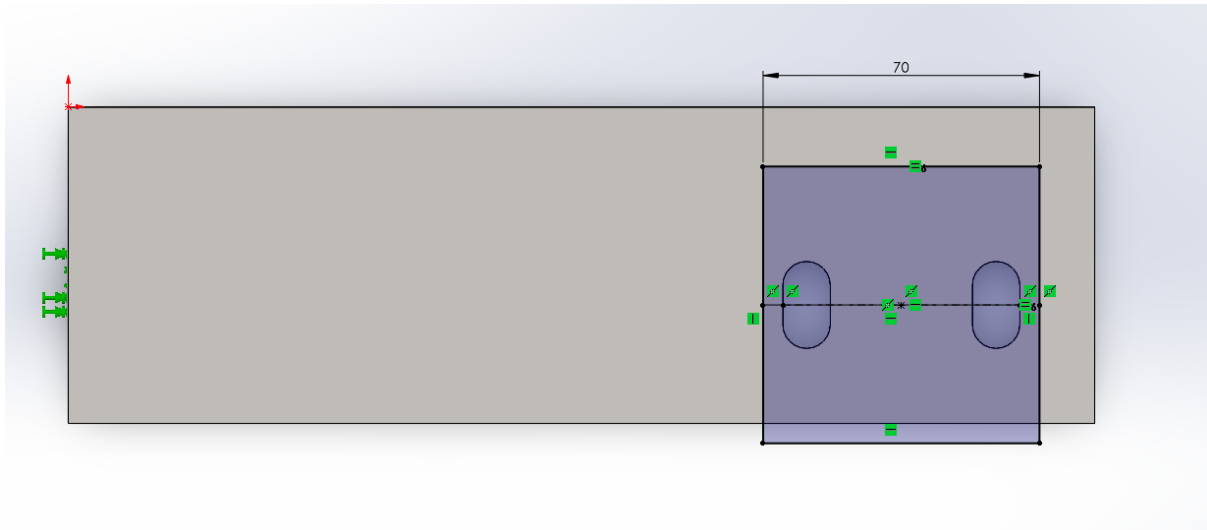


Figure 3 Force sketch

This sketch is then used along side a split feature to allocate the force distributed by the valve onto the support bracket, see figure 4 for force loading visual.

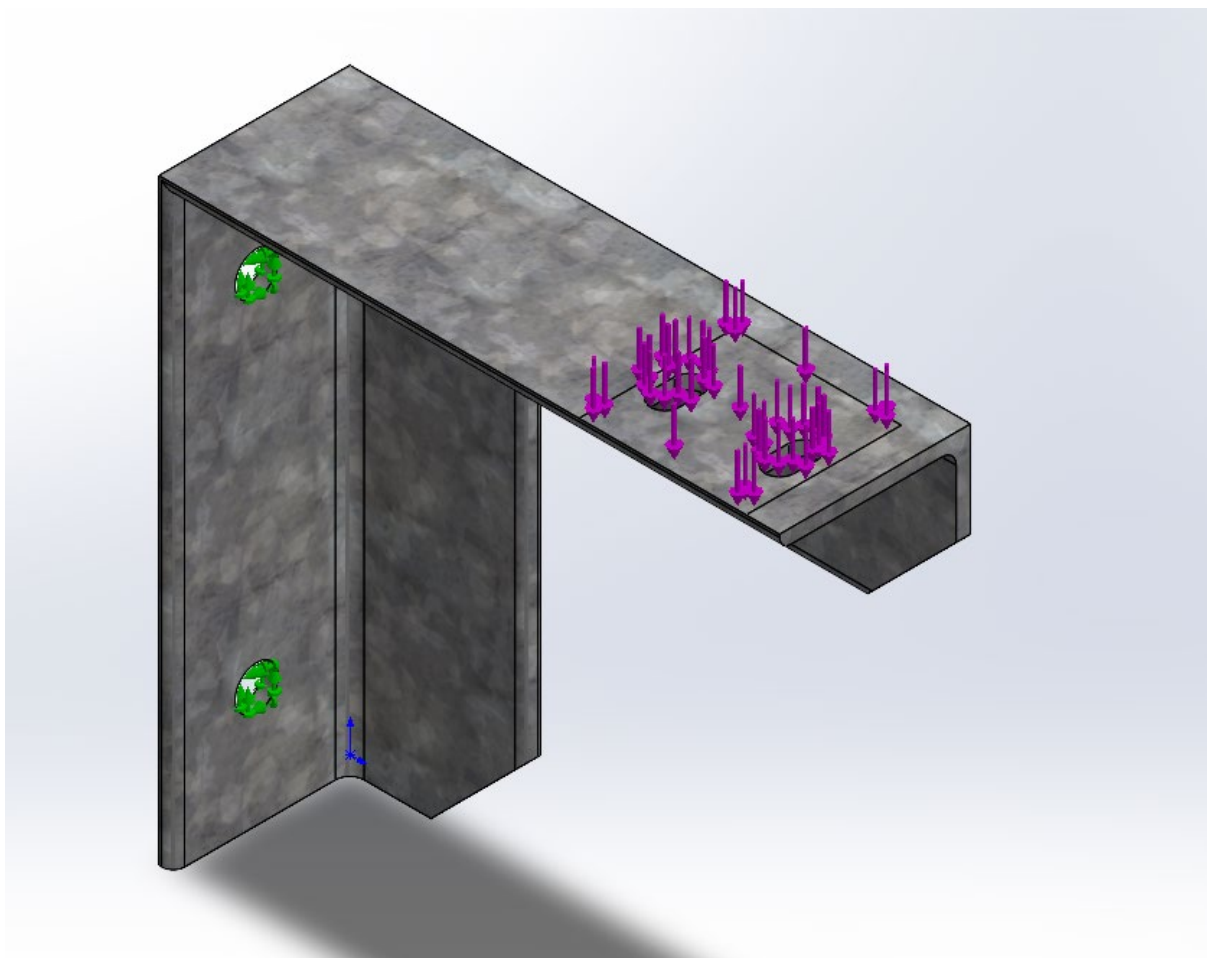


Figure 4 Static study force visual

A force of 216.4086N was used as this is the valve mass at 22.06Kg multiplied by earths gravitational constant at 9.81m/s^2 .

Lastly before running the simulation the mesh needs to be created, the mesh allows the software to break the component down into calculatable triangles across the face of the component. The mesh can play a key part in complex simulations as the more refined a mesh is the longer the study will take to calculate and therefore resolve with findings. This component has no complex geometry or complex shapes that need to be accounted for so a simple mesh can be calculated by SolidWorks shown in figure 5.

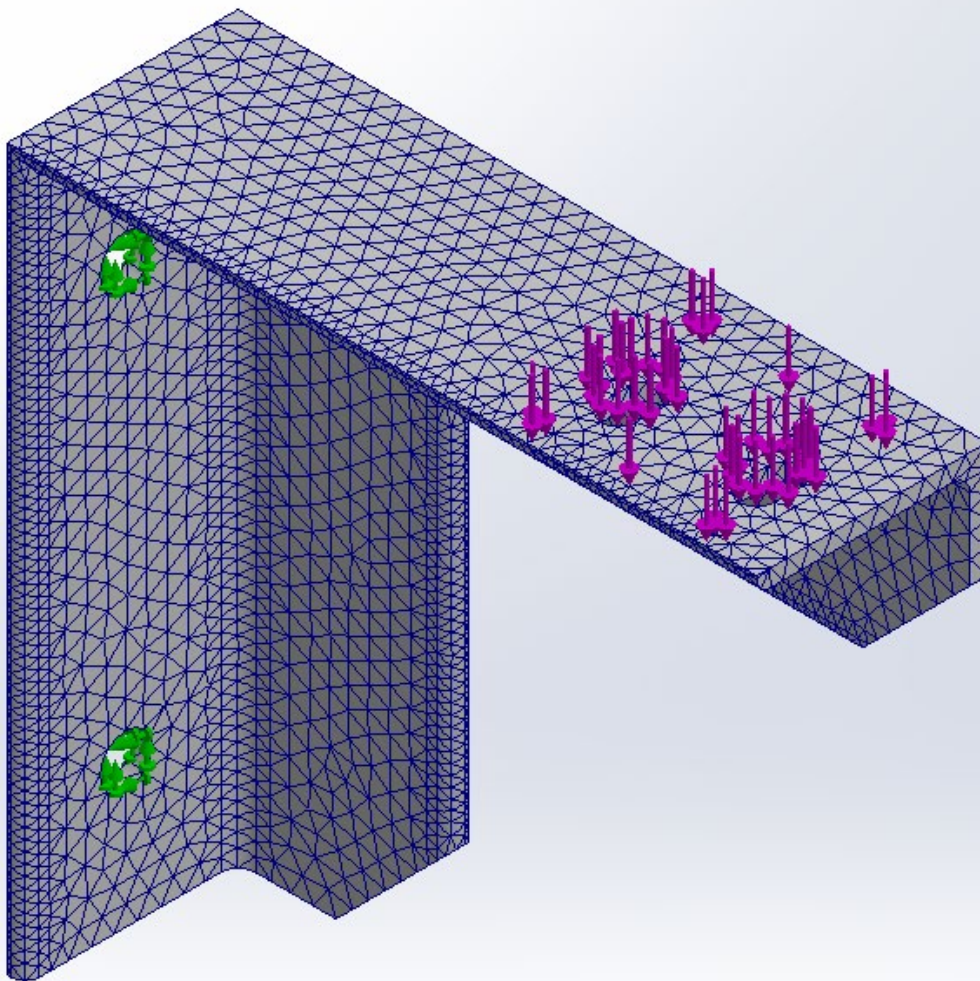


Figure 5 Static study mesh

Once the simulation had completed running the results can be viewed and interpreted. The first section to analyse is the stress within the component. A simple check to see

if the component will fail due to stress is to view the stress results visual and adjust the colour settings to show any areas that exceed the yield strength of the material in black, this will indicate any areas that have deformed into the plastic region and therefore will not return to their original geometry. Figure 6 shows this visual.

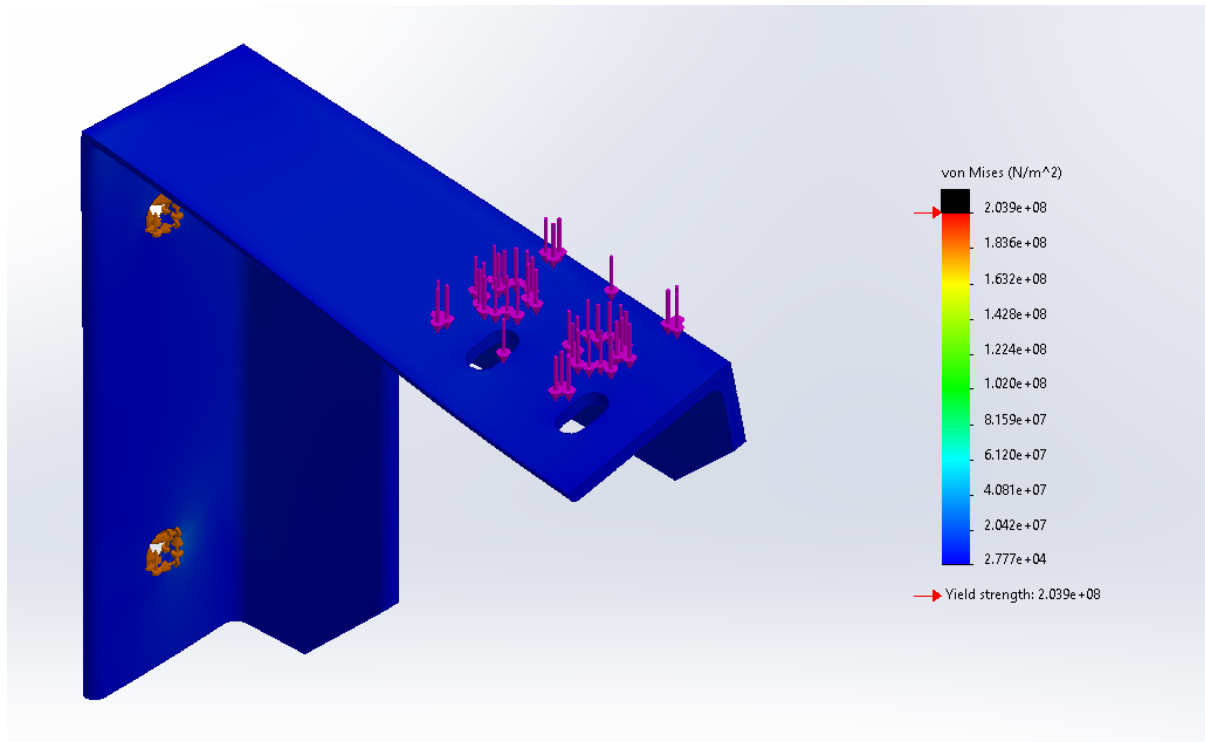


Figure 6 Yield Strength Stress results

As can be seen in figure 6 no part of this component exceeds the yield strength of the material due to normal loading from the valve.

Adjusting the visual settings to max out at the maximum stress achieved in the simulation shows us where the component is seeing and not seeing stress and therefore, we can assume where material can be removed. Figure 7 shows these results.

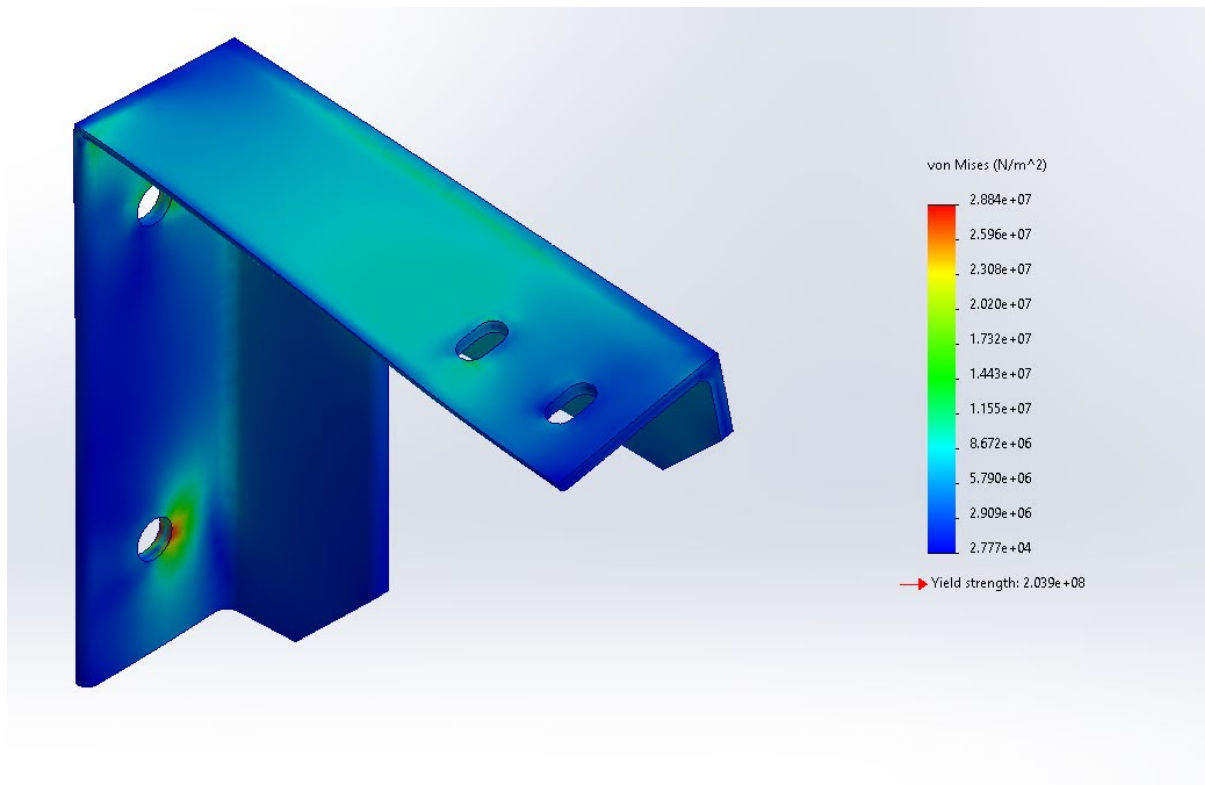


Figure 7 Static study Stress results

Looking at these results it can be obtained that the area seeing the most stress is around the wall bolting geometry and the top face to the left of the slots using figure 7 for orientation.

The amount of stress observed in the red areas is still well below the yield strength of the material, the maximum stress seen is $2.884 \times 10^7 \text{ N/m}^2$ and the yield strength of Galvanised steel is $2.039 \times 10^8 \text{ N/m}^2$, therefore it can be said that under normal loading the component only observes 14% of the overall yield strength.

Lastly the displacement needs to be analysed as one of the key features of this bracket is to take the weight of the valve and to hold it in place as the pipework assembly contains oxygen gas at high pressures and to avoid ignitions due to adiabatic compression and particle impact it is key to have all the components aligned to the best of the designer's ability. Having this in mind the designers do not want the support bracket to flex too much and cause the valve to be miss aligned.

The displacement observed can be seen in figure 8 below. The majority of the displacement is focused on the flat face directly under where the valve would be. This

is expected and when looking at the values the maximum displacement is 2.277×10^{-1} mm or 0.2277mm. this much deflection will not cause any issues for the pipework.

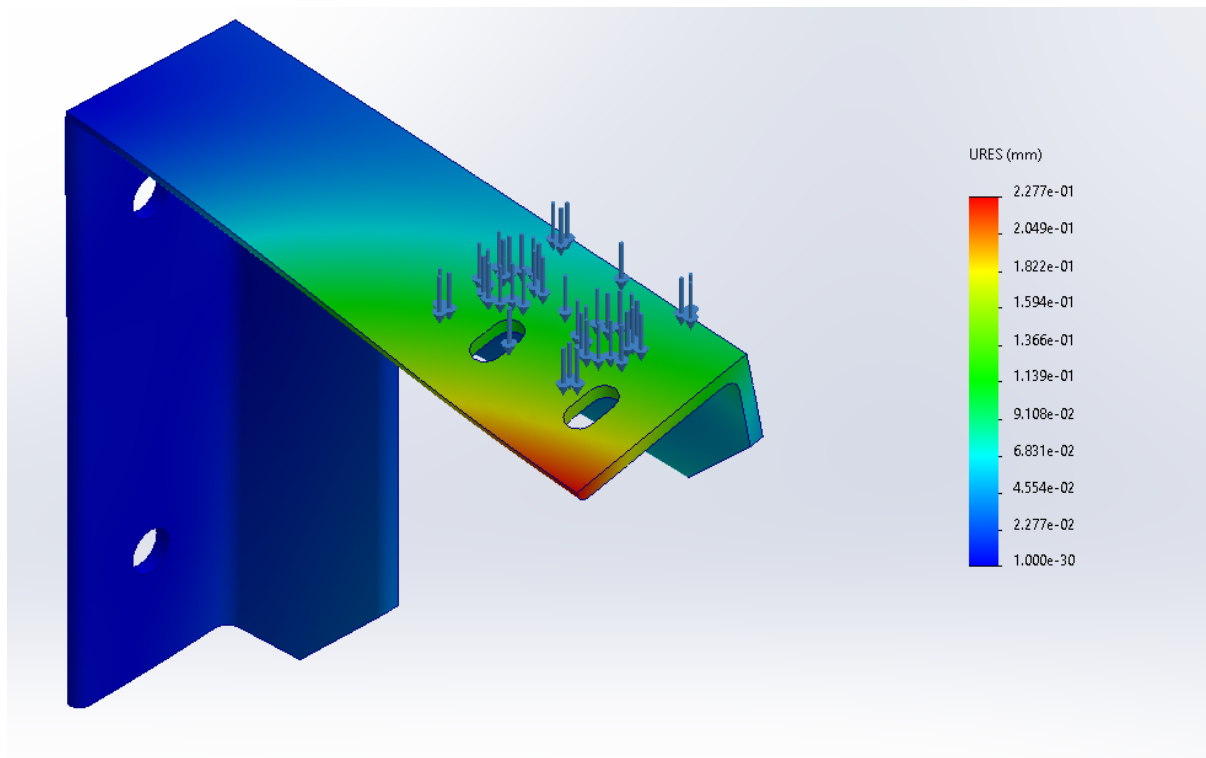


Figure 8 Static study displacement

After running this study, it was decided that another static study should be run with all the same parameters apart from the load should be increased from 216.4086N to 1000N to simulate roughly 5 times the weight of the valve that the bracket is designed to support.

The stress results can be seen below in figure 9. The area with the most stress is still concentrated on the lower bolt hole but it still far from the yield strength point of the material. With 1000N of force the maximum stress on the component is 1.333×10^8 N/mm² or 65% of the yield strength of galvanised steel. It should be noted that this is only in one area of the component, as can be seen in figure 9 most of the component is seeing much less stress.

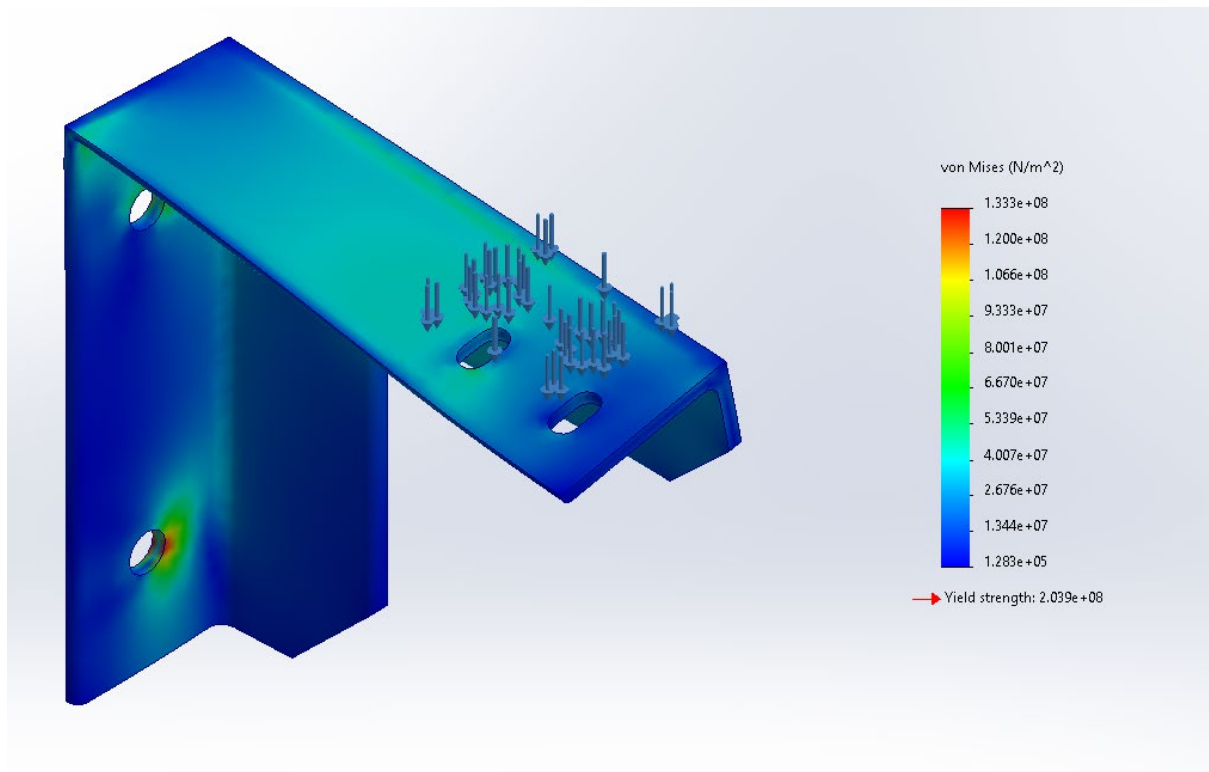


Figure 9 1000N Static study Stress results

The displacement results follow the same pattern as the displacement is in the same area as the 21Kg static study but with a larger value maxing out at 1.052mm. again this is concentrated in one specific area that can be seen in figure 10.

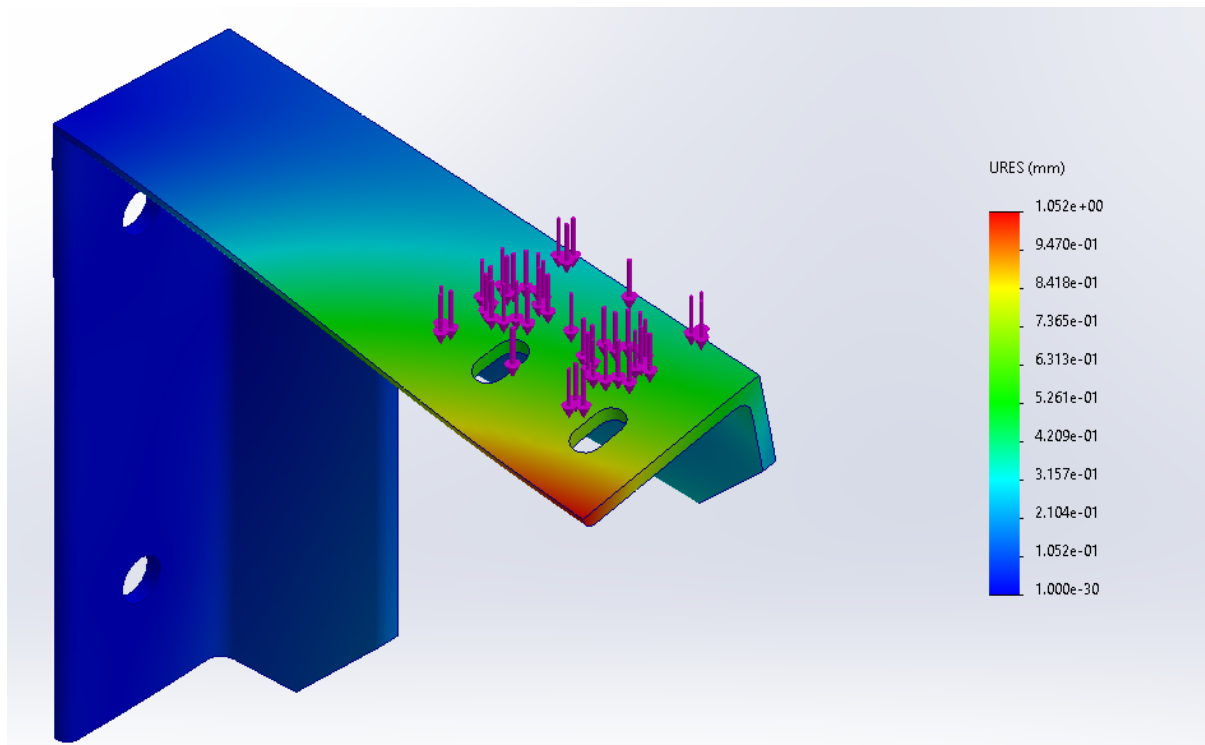


Figure 10 1000N Static study displacement results

These values and visuals can be used to compare the original design to any future designs that this assignment will produce.

2.4 Topology Optimisation

Topology optimisation is another process that can be preformed in SolidWorks 3D CAD with the Simulation add in. Topology optimisation allows engineers to take existing 3D models, apply loading via a force, constrain the geometry with areas that can and cannot be modified and lastly apply some pre-defined goal such as percentage mass reduction. SolidWorks can then run a series of numerical calculations and iterations to produce a meshed body that has the requirements inputted from the user. (Bendsoe, M.P. and Kikuchi, N, 1988)

Two preserved regions were assigned to the body. The first preserved region was the face that the valve rest on and is secured to. This includes to two slots. The other region selected was the bolting holes with a 10mm depth. This is where most of the stress was observed during the static study. These can be seen in figures 10 & 11 below.

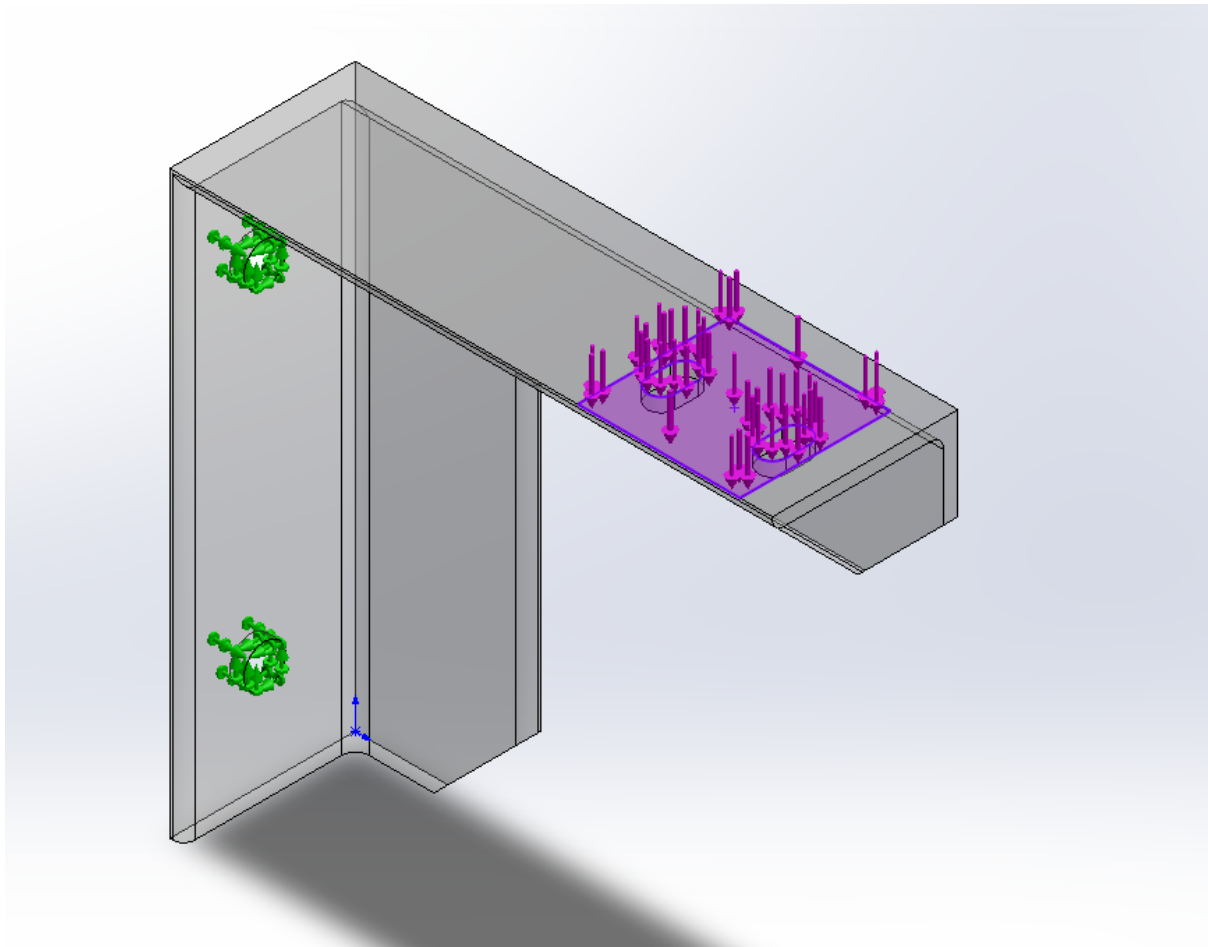


Figure 11 Topology Study preserved region 1

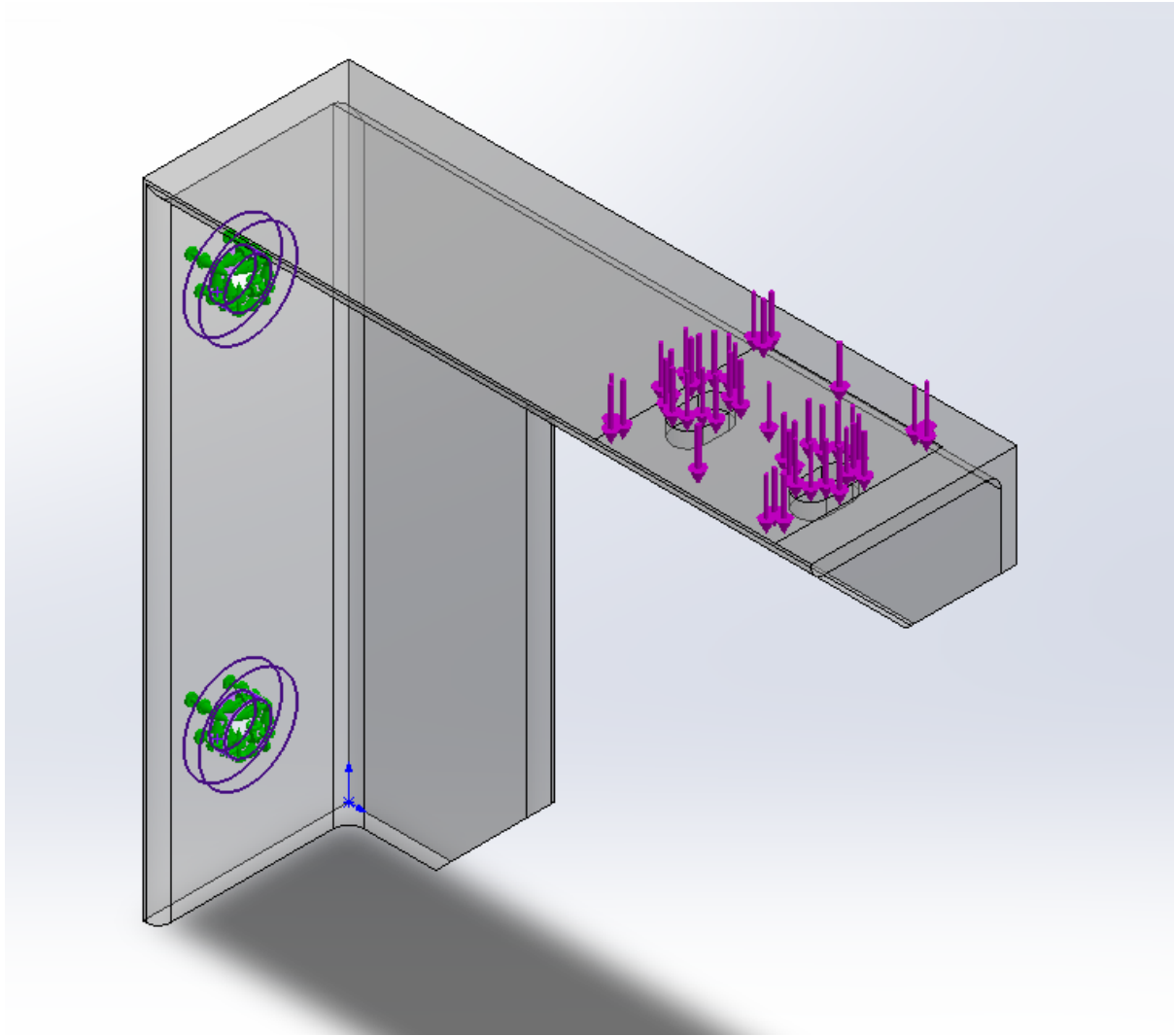


Figure 12 Topology study preserved region 2

The goals for the topology study were as follows, firstly a mass constraint was applied to reduce the mass of the whole body down to 1Kg from 3.26376 Kg.

The second goal is to minimise the deflection at the vertex shown in figure 13. This is where most of the displacement had occurred in the static studies. This is also where we want to minimise the deflection as movement here will directly impact the concentricity of the valve ports and pipework connections.

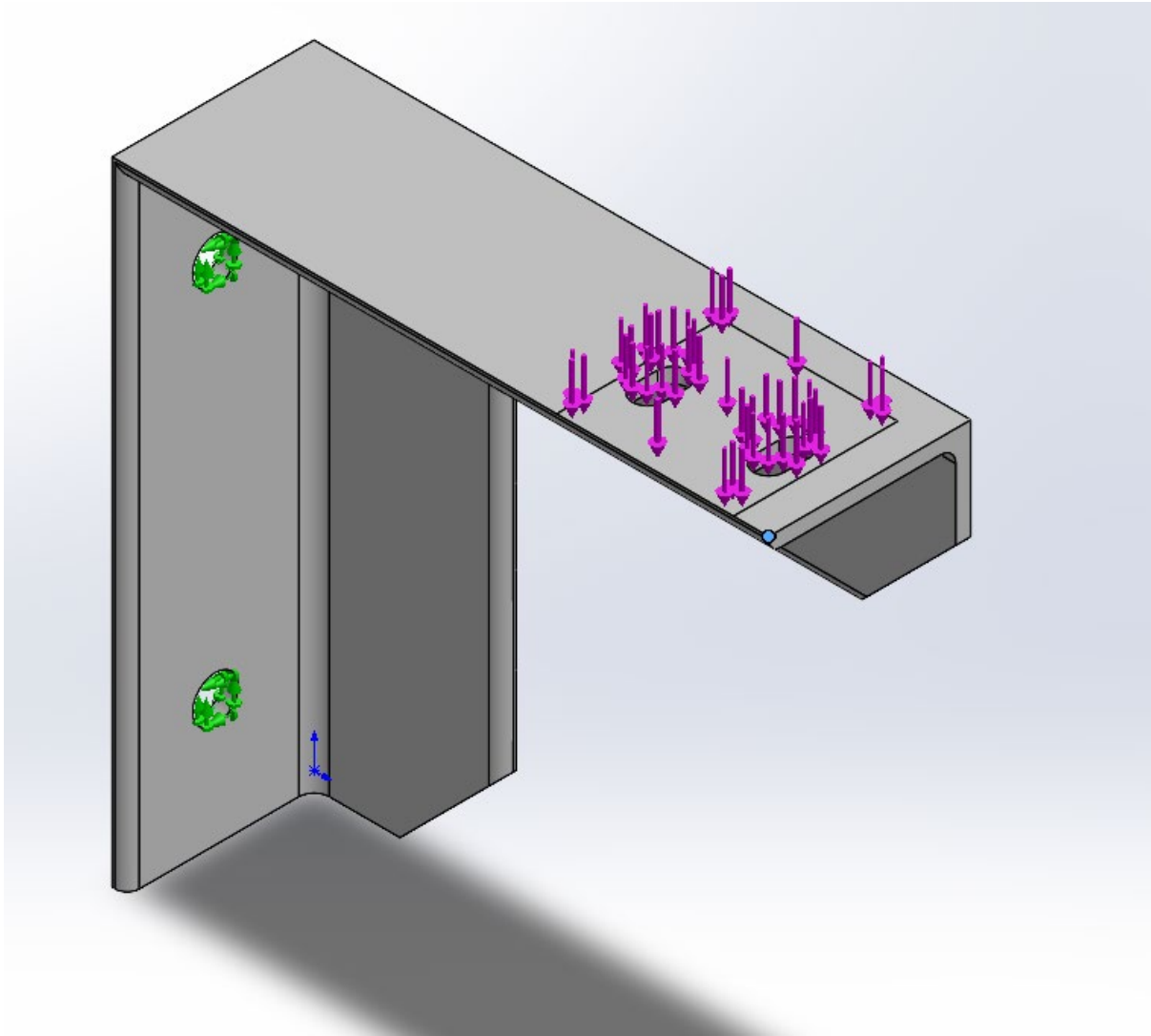


Figure 13 Topology study Vertex displacement marker

The final goal assigned to this study is to keep the maximum stress in the component to 0.75 of the yield strength of the material. This along side the 1000N load gives a large but not unrealistic safety factor for the final design of the bracket.

The topology study results show where material can be removed whilst retaining the structure required to hold up the valve. Figure 14 below is a graphic showing extreme removal of mass. This geometry can be exported and sent straight to the printer, for this design project we will be generating a new design using the results from the static FEA study and the topology study alongside DfAM principles.

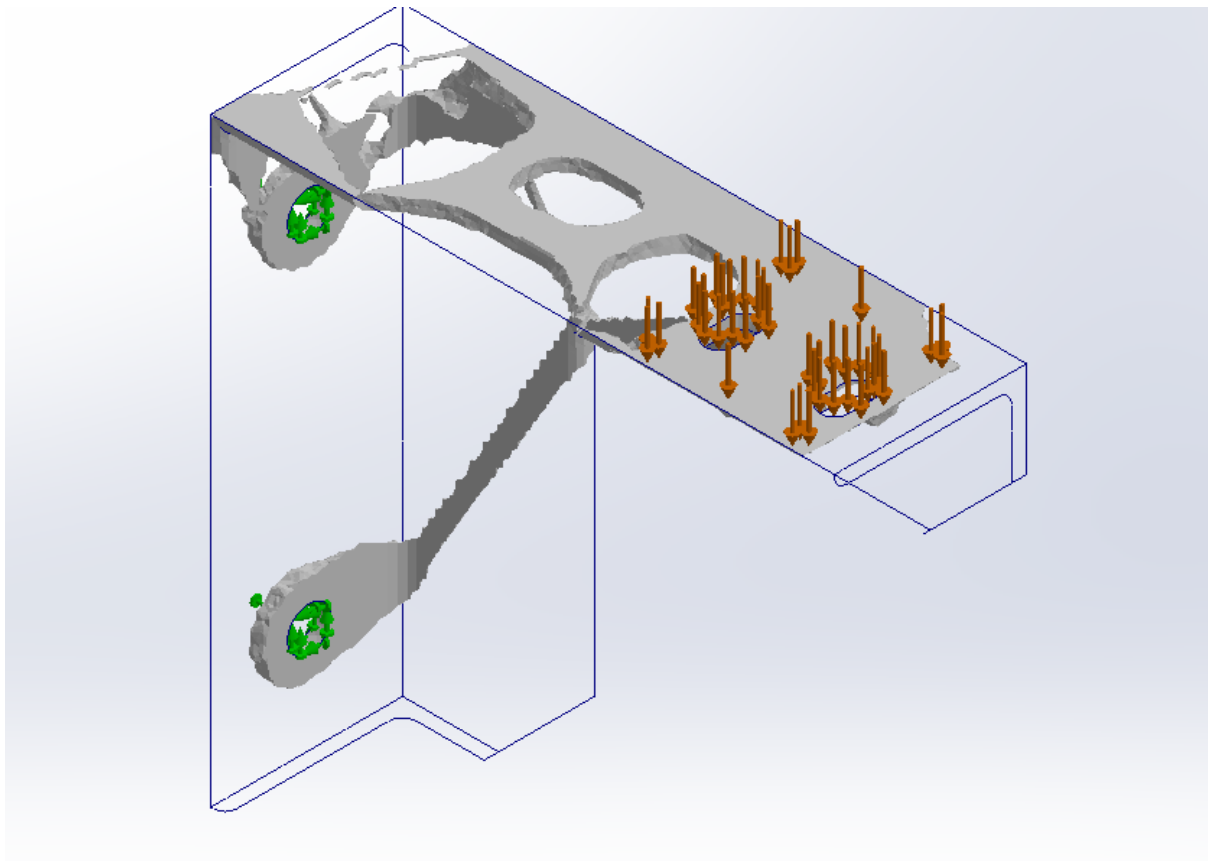


Figure 14 mass removal results

2.5 Design for Additive manufacturing

There are a lot of variables to think about when designing for additive manufacturing, the first area to dive into is the material choice, early on in this design study the choice to stick to a metal component was made, this is due to the location and use case for the product. The bracket is designed to be used in a fast pace high demand medical oxygen piping assembly. Minimising down time is a key design restraint, a good way to minimise down time to eliminate the worry of pipe supports failing over time. For this reason the choice to stick to a metal component has been made, the next step is to define the material. The part needs to be long wearing and non-corrosive. Galvanised steel was selected for the original component and for the new design Stainless steel has been selected. This is due to its anti-corrosion properties and the fact it is widely used in metal additive manufacturing.

The additive manufacturing technique used for Stainless Steel is Laser Powder Bed Fusion (LPBF). This process uses a laser to melt the cross section of a design very

accurately onto a bed of fine metal powder. The machine then adds a fine layer of powder on top and repeats the laser process. This builds up the design layer by layer. LPBF machines can produce very accurate components due to the use of lasers over filament nozzles such as in Fused Deposition Modelling (FDM) printing processes. (Additive Manufacturing Media, 2022)

The 3D printed part comes out of the machine fused to the build bed and with any supporting structures that the slicing software has generated also fused to it. This output is then put into a heat-treating oven to relieve any internal stresses caused during the printing process. Once the part has gone through the stress relieving process it can be removed from the build place and the supports can be removed via CNC machines, the CNC machines can also add any features such as spot facing or threading holes. The surface finish of the part may need touching up depending on the design requirements.

Slicing software is a program that converts the 3D model into the individual layers that the 3D printer will produce, it will also calculate where any support structures are required. Within the slicing software you can select variables such as infill and wall thickness. For prototypes tweaking these variables can produce quicker samples but the samples may have loose strength so it is key to weight up and decide the use for the sample before adjusting these parameters. (AdditiveX, 2023)

2.6 Final Design

The final design to be Printed using a LPBF printer and stainless-steel powder can be seen in figure 15. The design has removed a large amount of mass from the lower section of the bracket whilst retaining the location and structure around the bolting holes. The topology study revealed that a lot of the material can be removed if a branch is kept in the relevant areas to retain rigidity and strength.

The isometric, front, side and top view for the new design can be seen in figures 15 – 18 below.

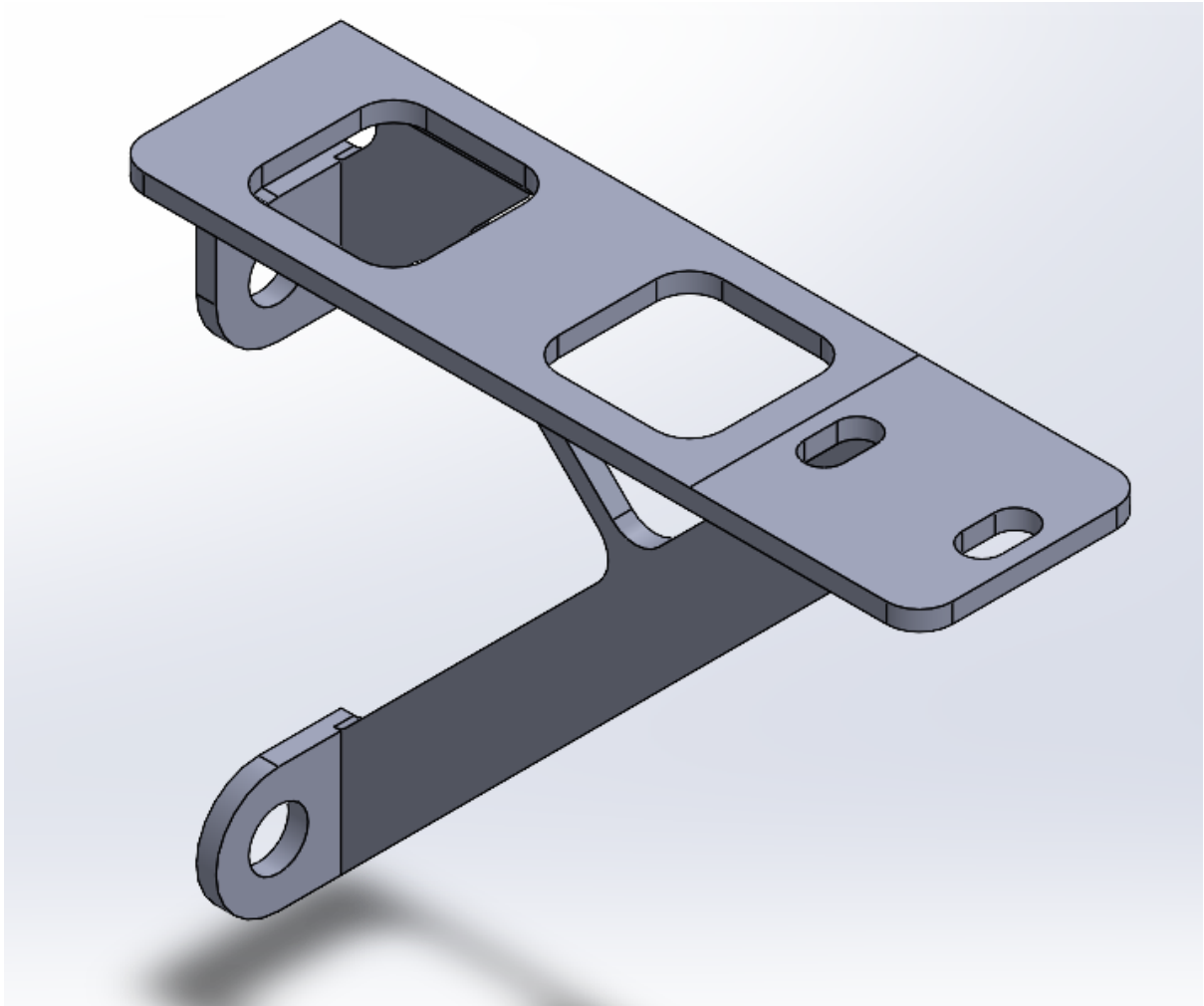


Figure 15 Isometric View Final Design

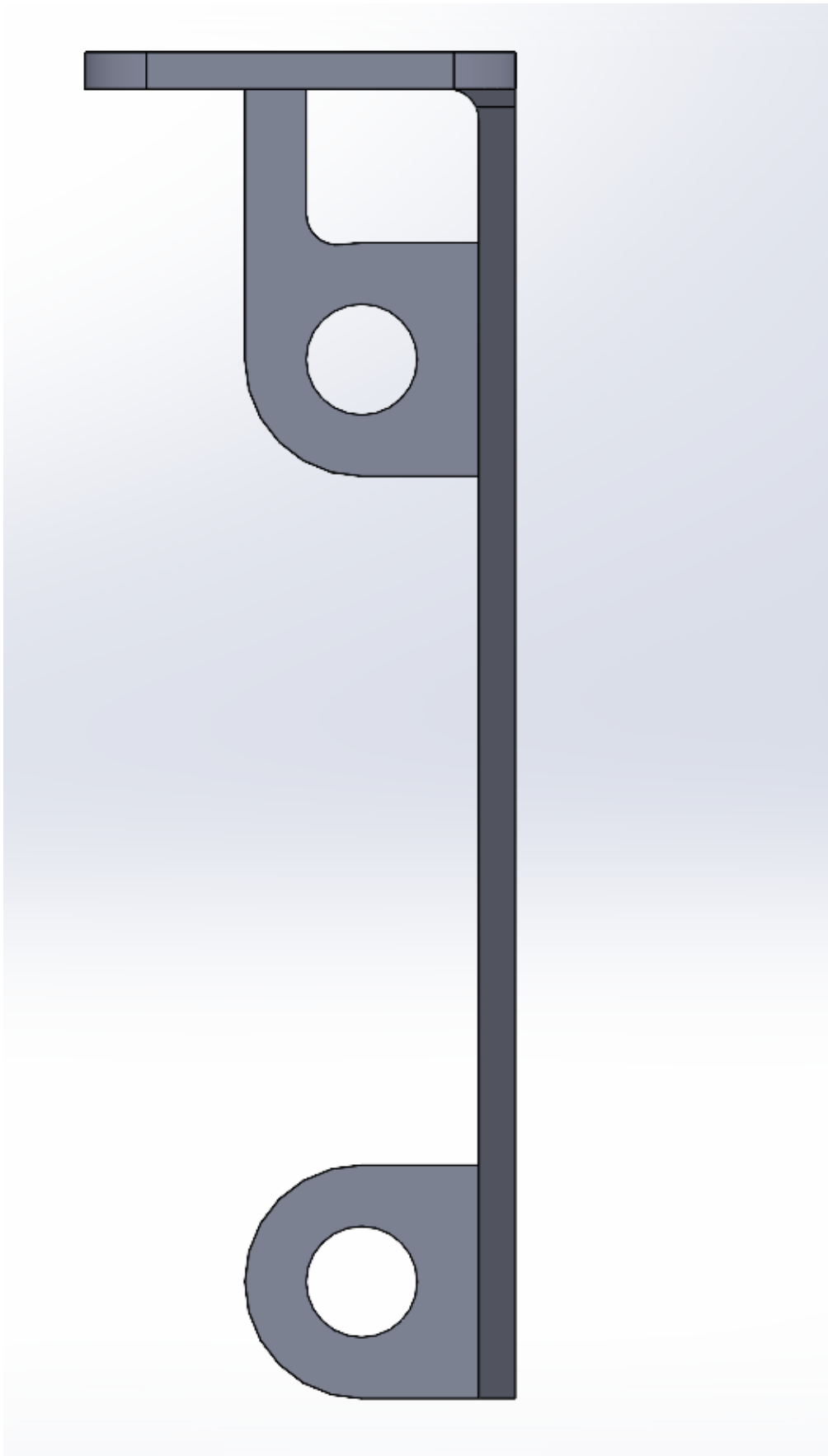


Figure 16 Front View Final Design

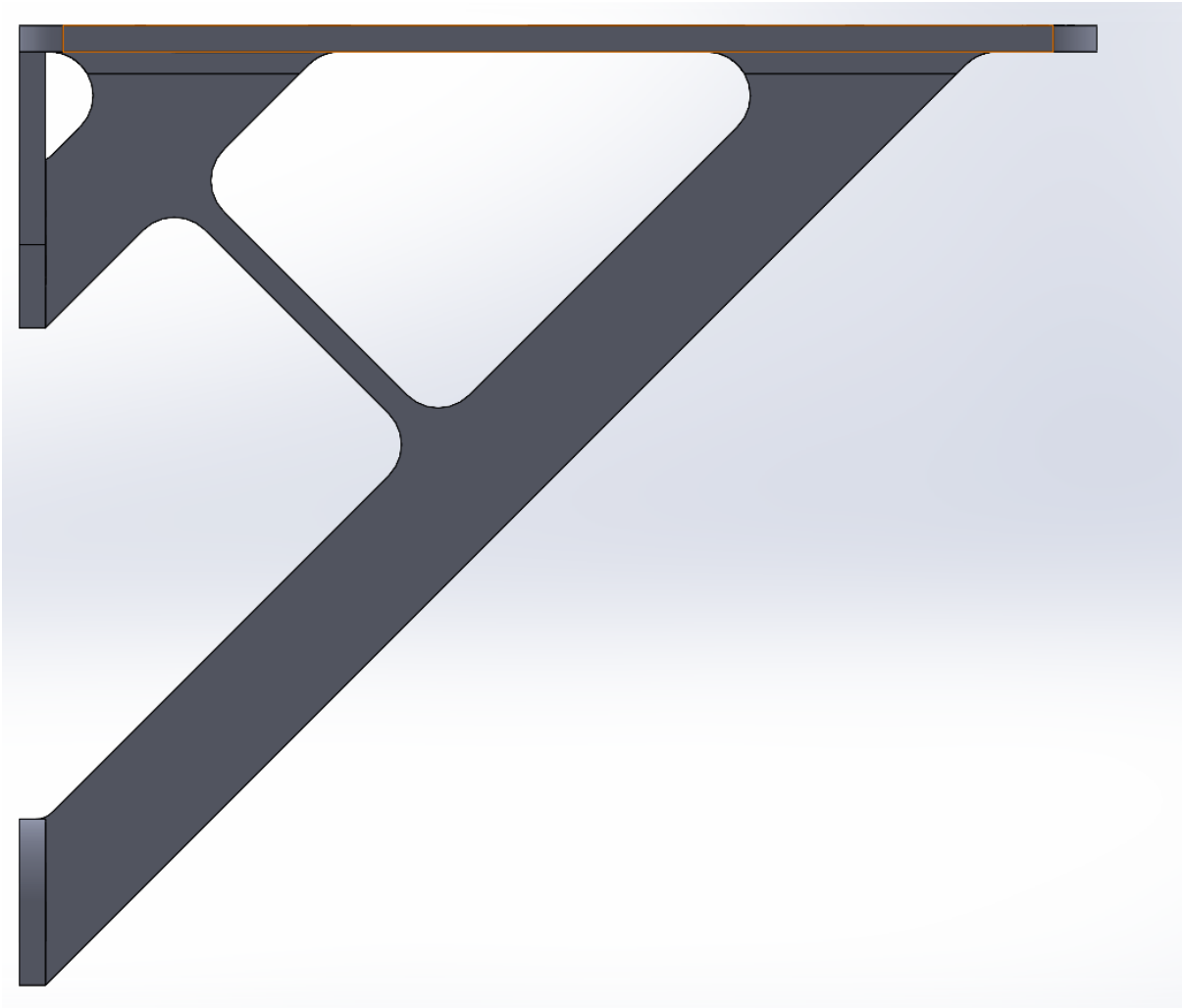


Figure 17 Side View Final Design

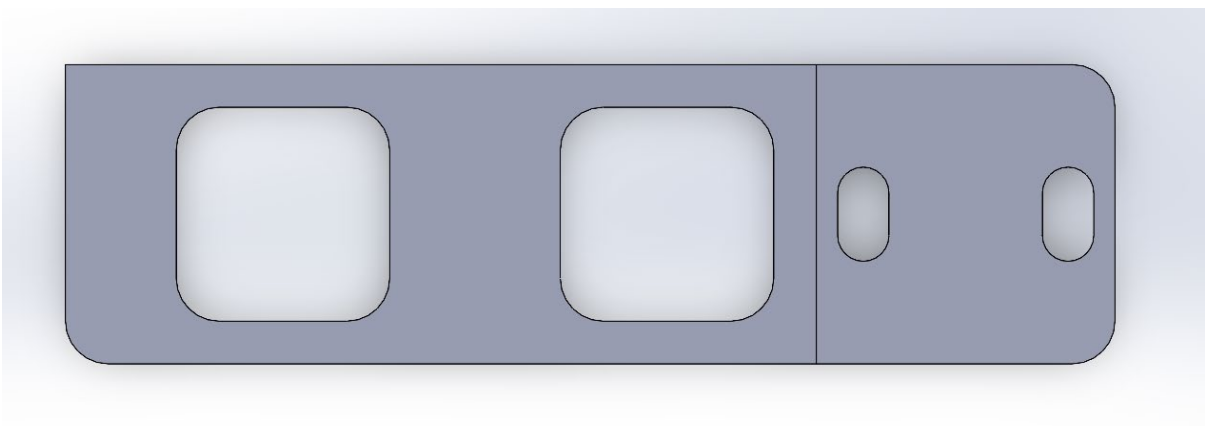


Figure 18 Top View Final Design

Another static study FEA simulation has been run on this final design and the results can be seen below in figures 19 – 20. Once again a force of 100N was used along with the same bolting hole fixtures.

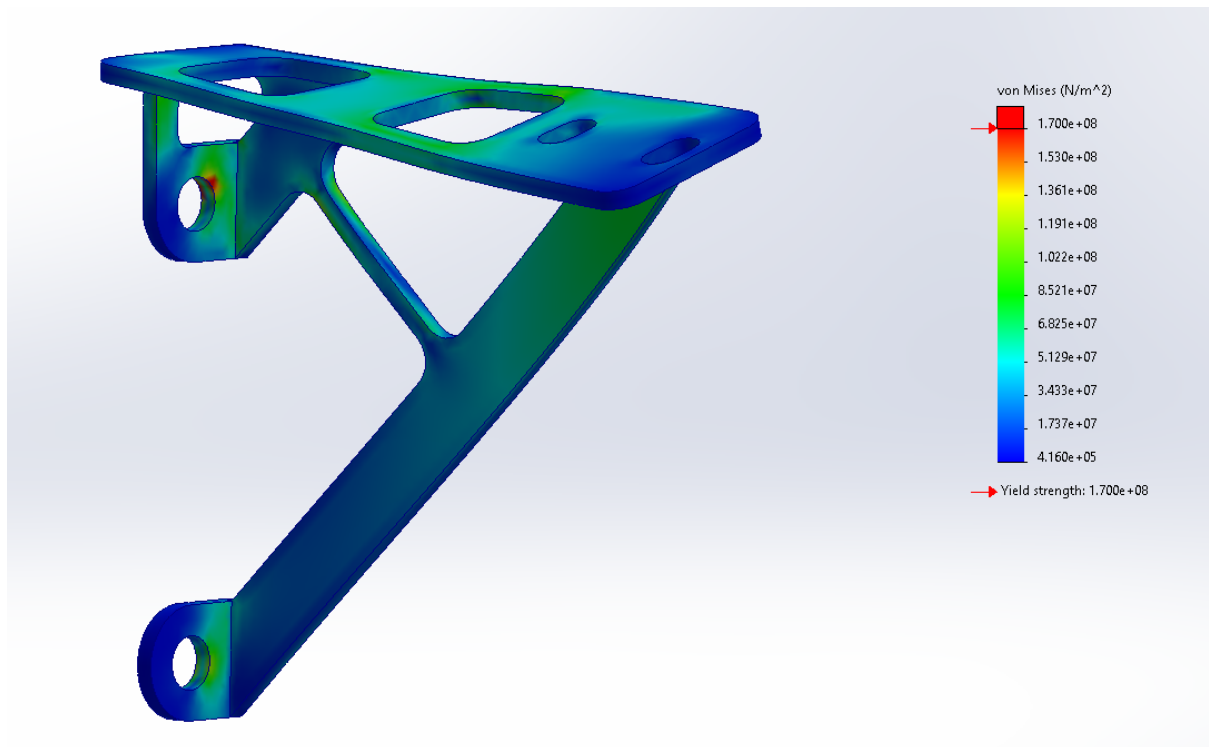


Figure 19 Final Design Stress Results

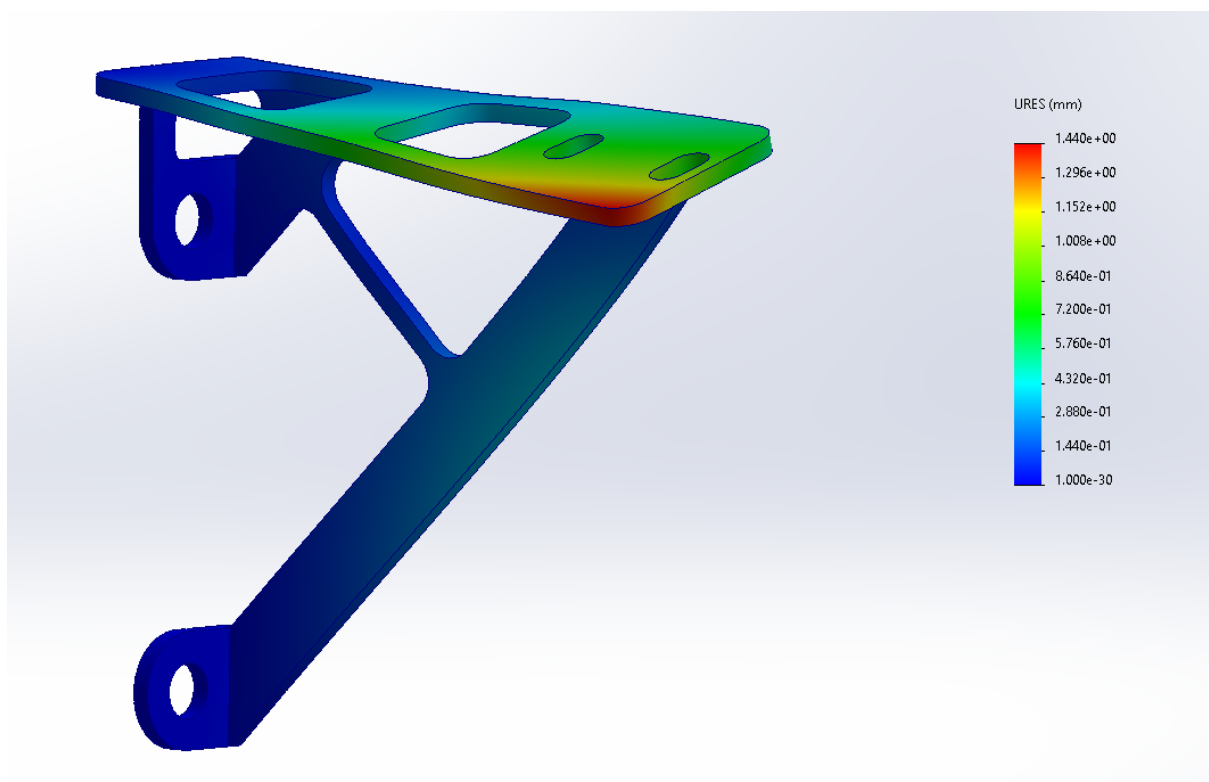


Figure 20 Final Design Displacement Results

The only area seeing high amounts of stress is in the same place as the original design and this is around the upper bolting hole. This is slightly over the yield strength of the

material but with the extra force being applied in the simulation the bracket should be fine in normal operation.

The displacement has maxed out at 1.44mm on the simulation.

If more time was available extra simulations could be run and more in-depth analysis of the design could be conducted. The steps show in this task have shown the understanding of FEA, DfAM and topology optimisation in CAD systems.

3.0 Task 2 – High-volume production – Portable Wind Turbine Redesign

To update this design for mass production we need to look at where the design may be over complicated or unnecessary steps of the assembly/ production process can be removed.

The key function of the current design are that the wind turbine is portable due to its foldability, we do not want to lose this feature as it is a good Unique Selling Point (USP).

On initial review of the design there are multiple instances where multiple components could be consolidated into one machinable component, this would save time during assembly and reduce the production time due to tooling requirements. A good example can be seen in figure 21 below, the items in silver and yellow have been designed to be 3D printed but when going for mass production these could be fused together and either machined or injection moulded.

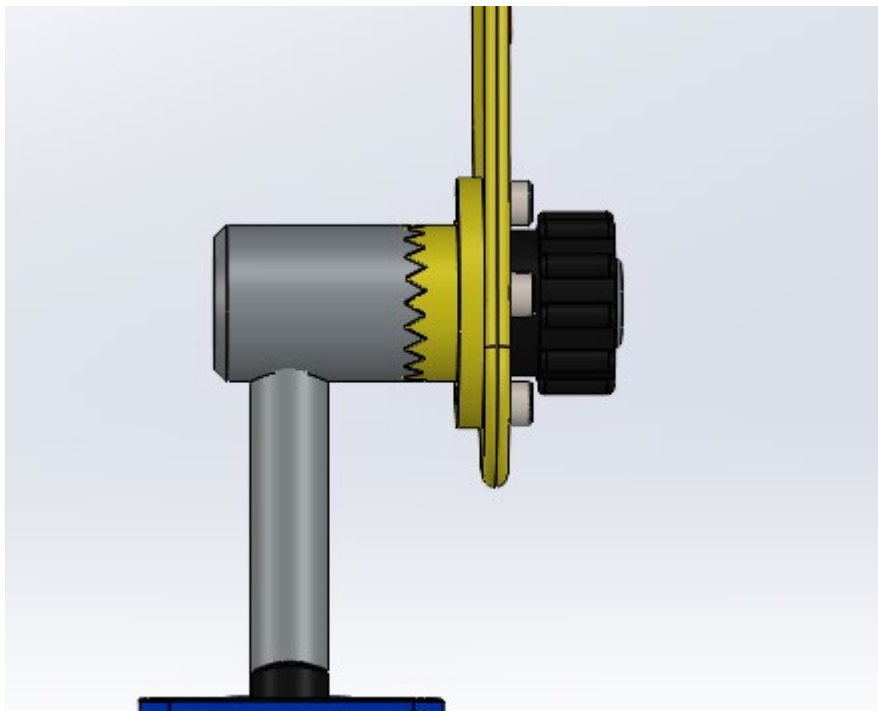


Figure 21 Component consolidation

The same logic can be imposed on the propeller and hinge feature. They can be consolidated whilst keeping the foldability of the overall assembly (see figure 22).

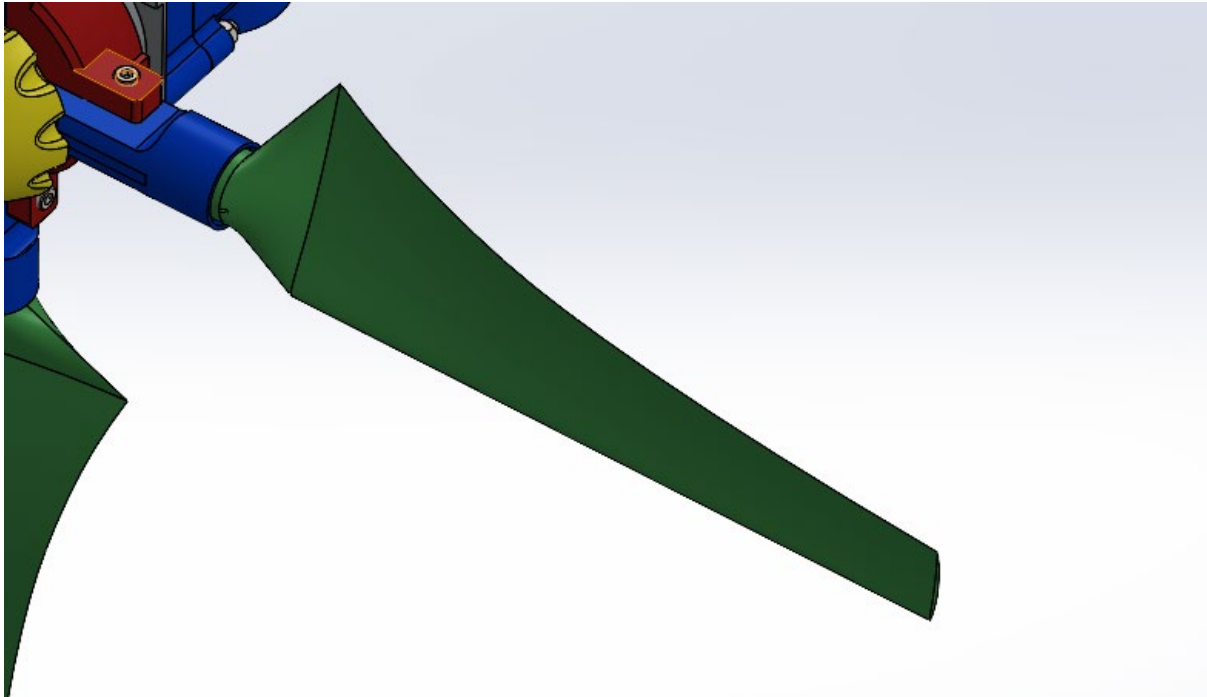


Figure 22 Consolidation 2

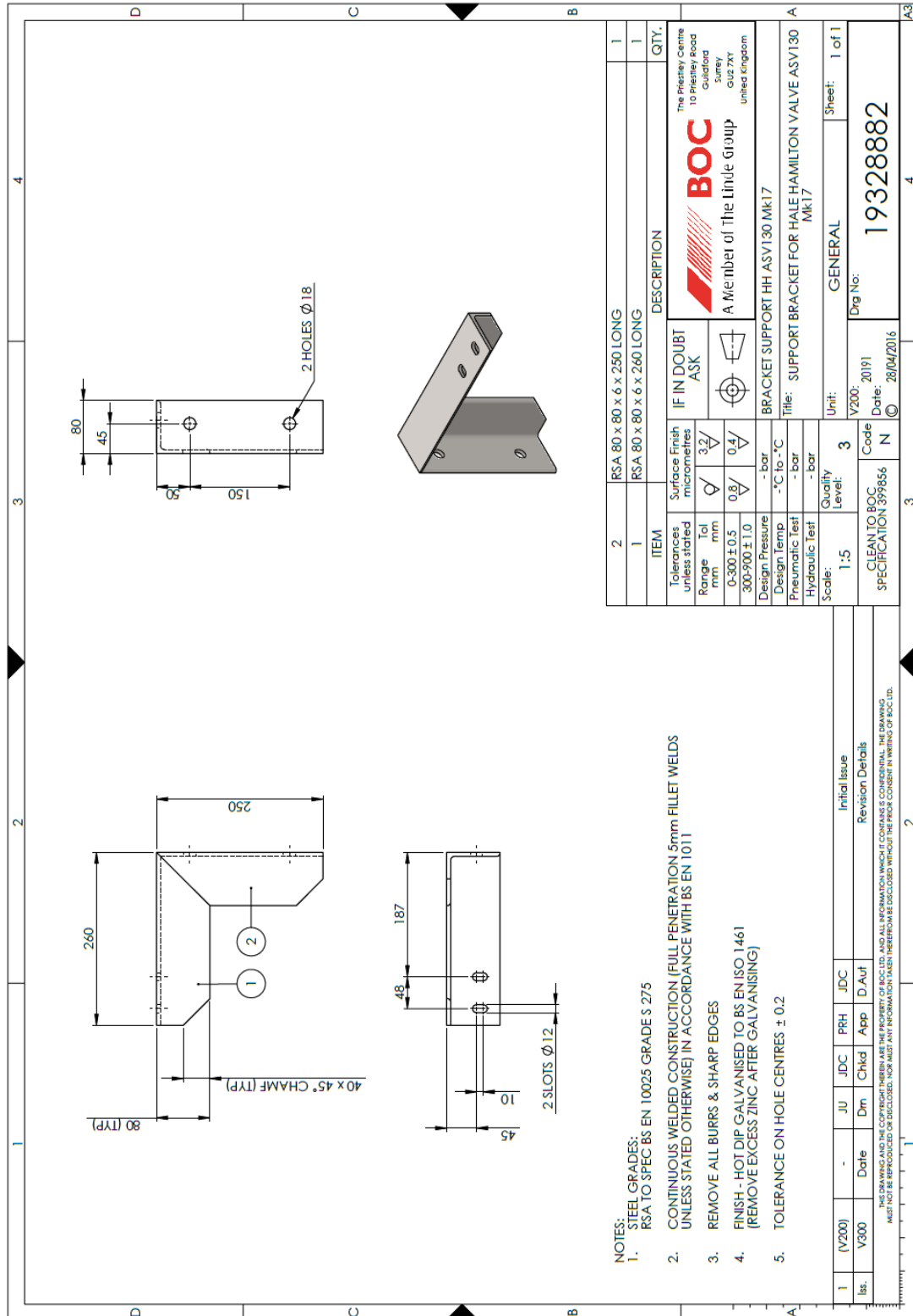
Another area to think about during mass production is assembly, the assembly process is a key area where time and money can be saved. A smart product can be easily assembled. This can be achieved by designing the components to be assembled using a top-down process where all the components fit on the same axis.

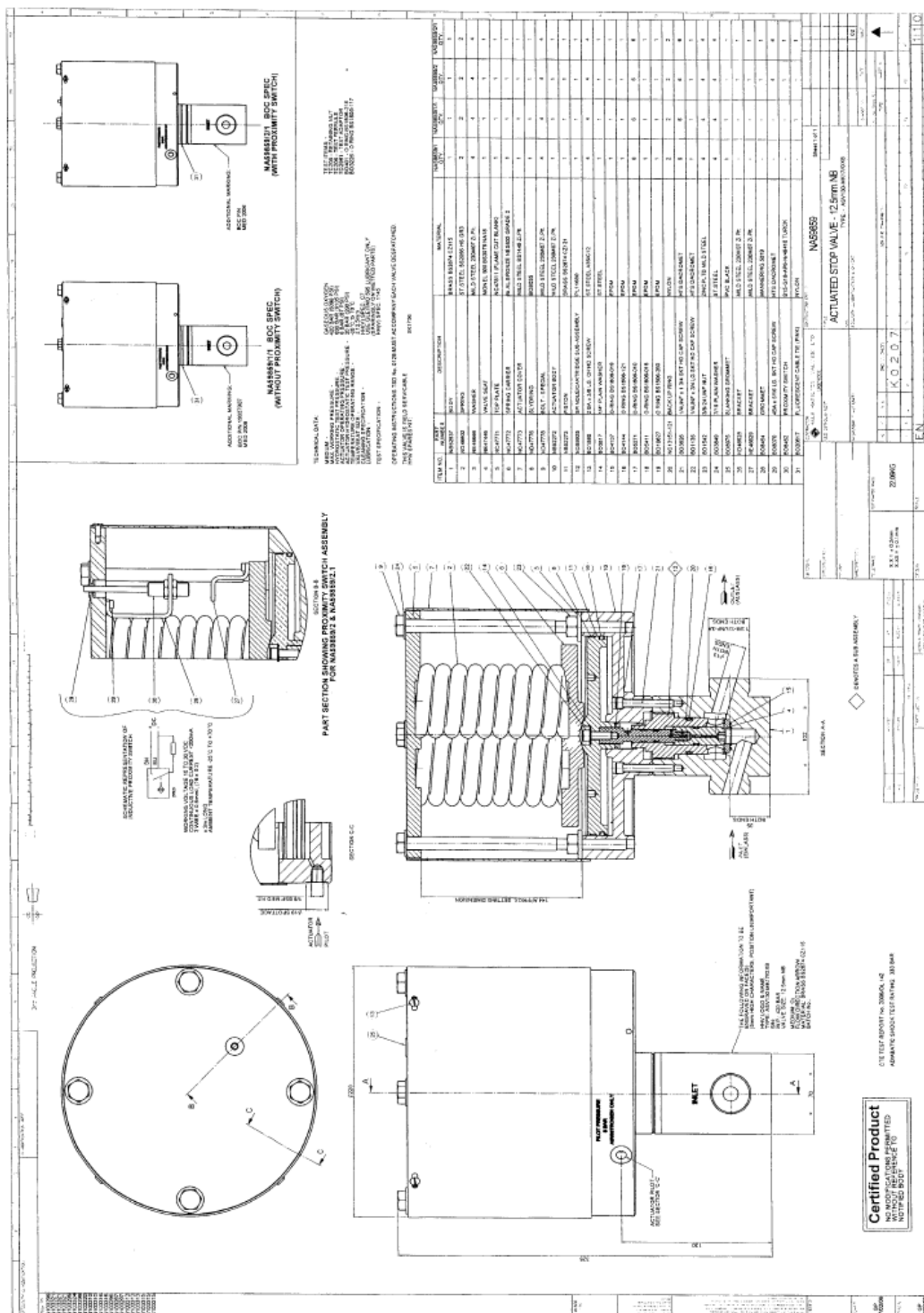
In the current design there are a lot of bolts in various planes and axis, along with the previous mention consolidation and top-down assembly this design would be simpler and quicker to assemble.

Lastly a good area to consider is the joints, all of the joints/ connection in the current design are bolts and nuts, the use of snap connections can speed up the assembly process.

Appendix

Appendix 1





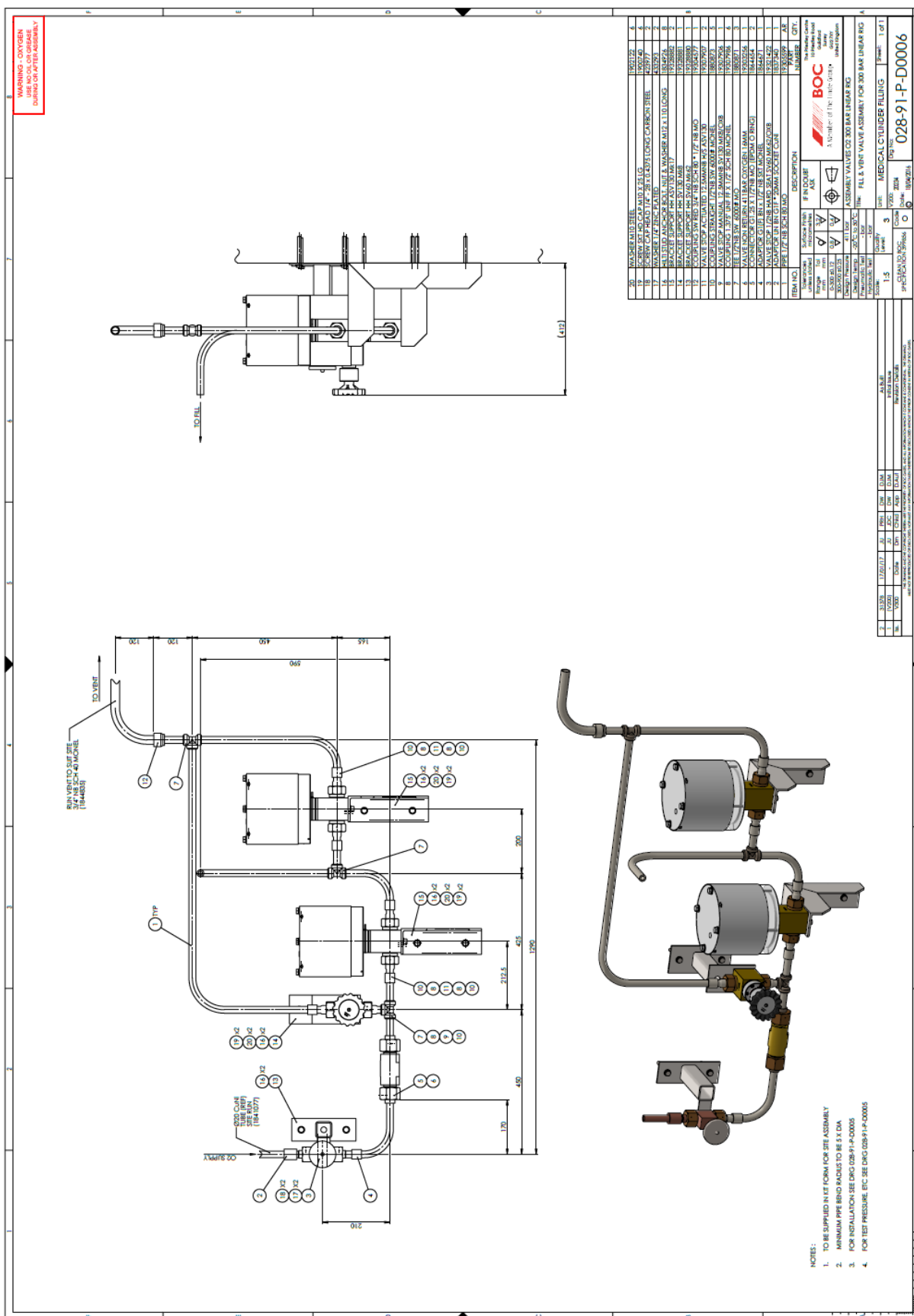
BOC GASES
VENDOR DRAWING

V200	V300
14770	

BOC Ref. 19307907VA-1-2 Issue 1
Vendor Name HALE HAMILTON VALVES
Vendor Dwg No. NA 59859 Issue 1
DA Review DJ MaHeard Date 23/05/06

Review of any drawing/specification by BOC shall not relieve the supplier of any liability or responsibility for the design or goods manufactured by the supplier in accordance therewith.

Appendix 3



References

Additive Manufacturing Media (2022), What is Laser Powder Bed Fusion? , available at: https://www.youtube.com/watch?v=r8_M995cwAA (Accessed 02-07-2023)

AdditiveX (2023), What is Slicing Software, and what does it do? Available at: <https://additive-x.com/blog/what-is-slicing-software-and-what-does-it-do/> (Accessed: 03/07/2023)

Bendsoe, M.P. and Kikuchi, N. (1988) Generating Optimal Topologies in Structural Design Using a Homogenization Method. Computer Methods in Applied Mechanics and Engineering, 71, 197-224.

FABORY (2023), Hot Dip Galvanizing, available at https://www.fabory.com/en_GB/knowledge_center/technical/surface_treatments/hot_dip_galvanizing (Accessed: 01/06/2023)

SolidWorks (2023), SolidWorks 3D CAD, available at <https://www.solidworks.com/product/solidworks-3d-cad> (Accessed 28/05/2023)