Design Description

The goal of this project is to design an L-shaped bracket. The bracket is to be made of annealed steel with the following parameters:

Parameter	Description	Value
σ_Y	Yield Stress	$200\ MPa$
E	Modulus of Elasticity	$207\ GPa$
υ	Poisson's Ratio	0.27
ρ	Mass Density	$7860\ \frac{kg}{m^3}$

The bracket is subject to a distributed surface load with a total force (F) of 40 KN. The bracket is on fixed support on the top left corner. The bracket may not exceed the following geometric dimensions shown in Figure 1 apart from a small fillet.

The goal of the project will be to optimize the shape of the bracket such that the weight of the bracket is minimized. Furthermore, the bracket should not fail under load, or plastically deform.

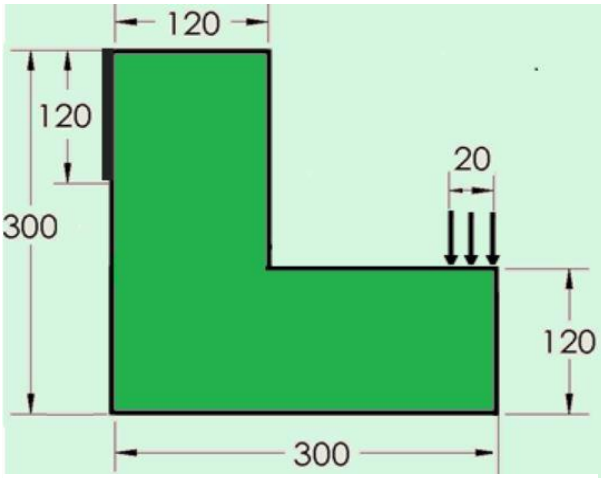
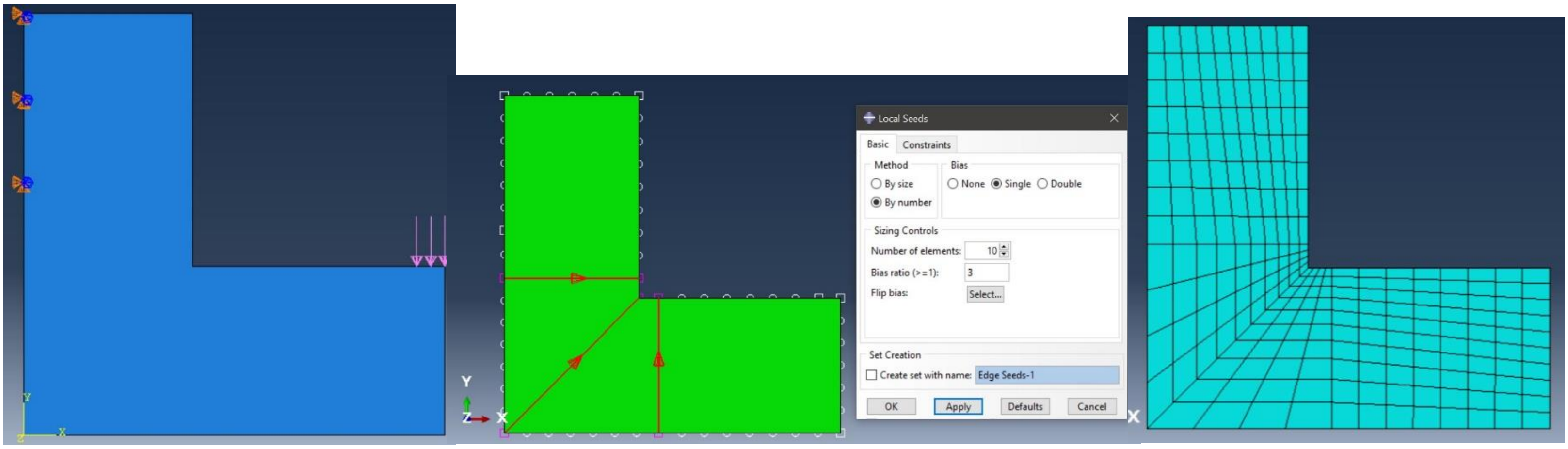


Figure 1. Geometric Constraints. All dimensions in mm. The thickness of the bracket is $0.050\ mm$.

ABAQUS Model

A part was created with the maximum dimensions specified in the problem statement. The mass was measured using Abaqus mass properties: 2.26 kg. Boundary conditions and loading conditions were applied. The partition was made around the suspected stress concentration point. Global seed size was assigned to 20 with a bias ratio of 3 and number of elements per seed of 10. The following mesh was created.



Method: ○ By size ● By number

Bias: ○ None ● Single ○ Double

Sizing Controls

Number of elements: 10

Bias ratio (>=1): 3

Flip bias: Select...

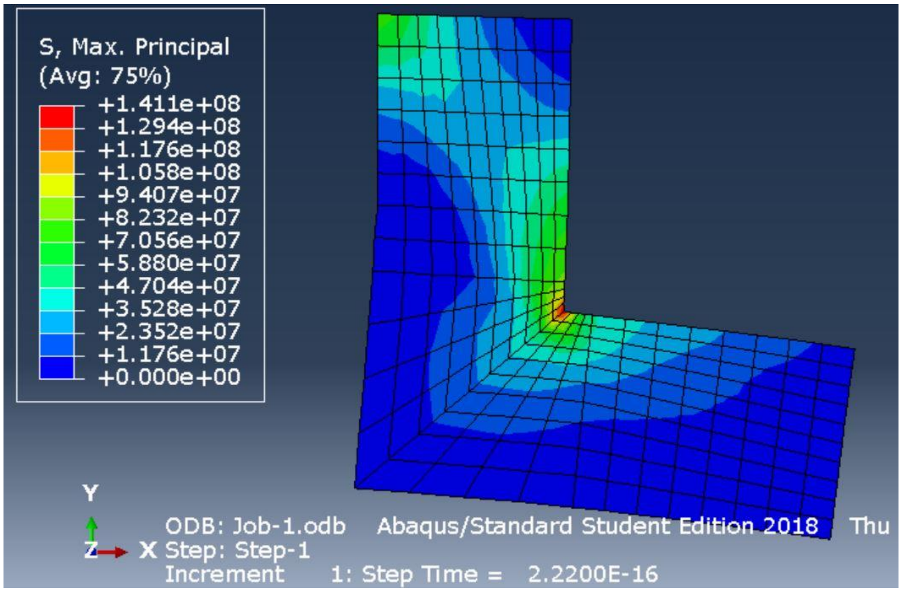
Set Creation

☐ Create set with name: Edge Seeds-1

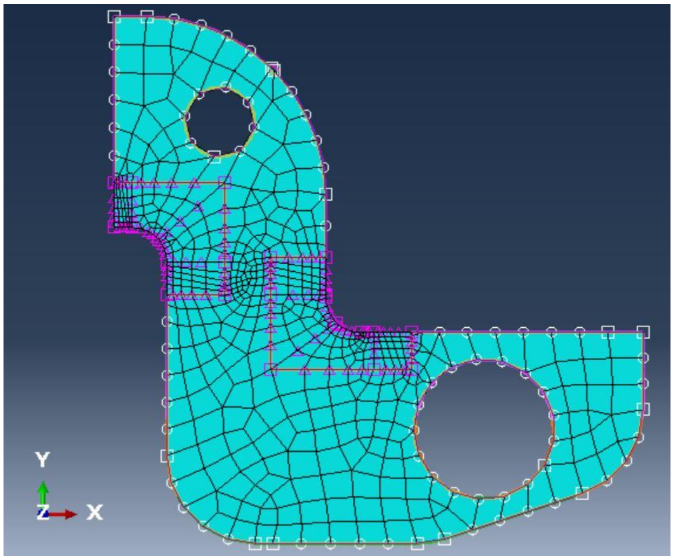
Initial Simulation

The FEA simulation shows a maximum principle stress under the Yield Strength of the material. This indicates no plastic deformation within the material and proves that the part doesn't fail under these conditions.

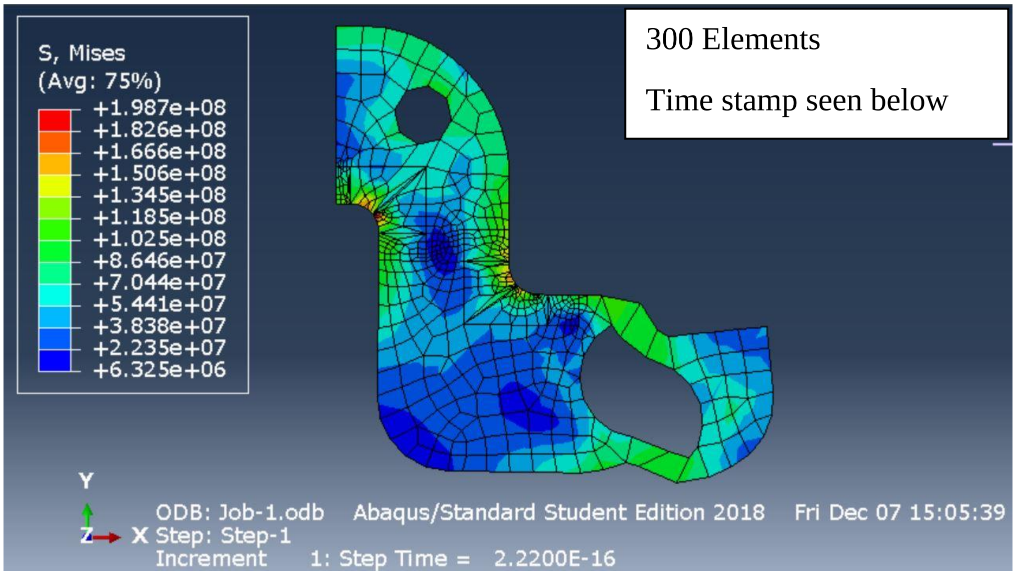
The concentration point of stress is at the corner point location as expected. The indefinitely varying stress at this location indicated the presence of a singularity point.



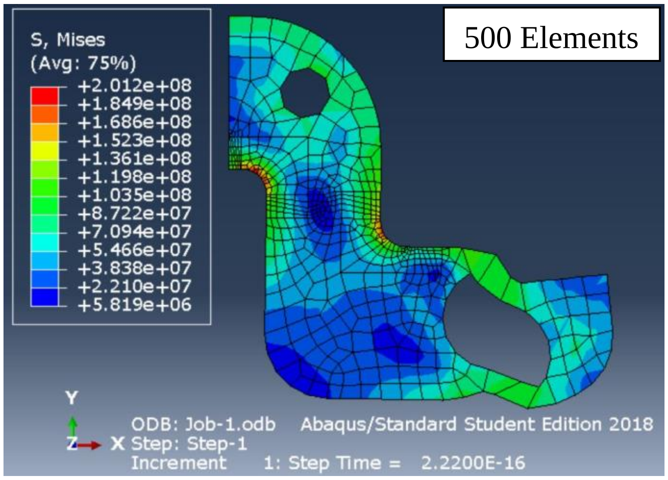
Final Design and Simulation

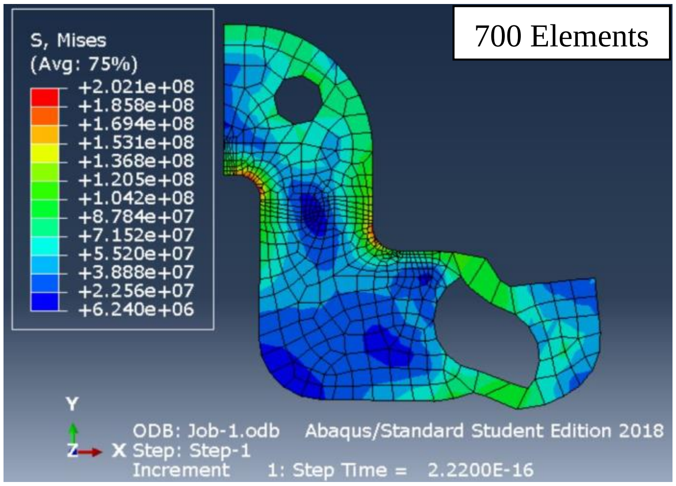


The following changes were made to the design and the part was again meshed but this time with partitions radiating towards each stress concentration points. The simulation was run with three iterations. The first, second, and third had 300 total elements, 500 total elements, and 700 total elements, respectively. More specifically, each successive iteration increased the number elements per local seed. As a result, the partition edges that radiate towards the stress concentration were meshed with increasing fineness. This was desirable because these stress concentrations are of highest concern in guarding against plastic deformation. As seen below, the maximum stress in the part converged to a value below the Yield Strength.


300 Elements

Time stamp seen below


500 Elements


700 Elements

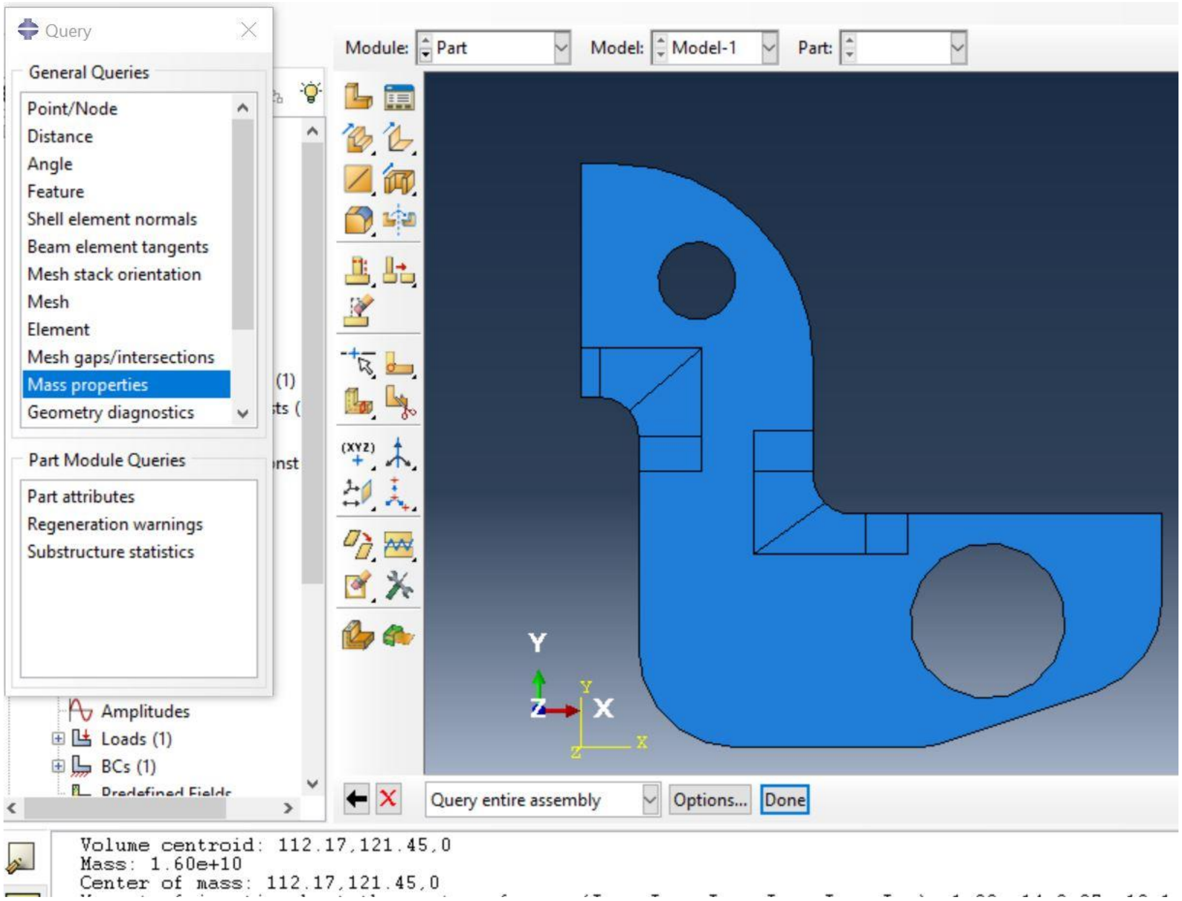
Discussion

The design of the bracket was re-engineered with the goal of improving its structural integrity while minimizing the mass of material used to create the bracket. This was done by a few design changes. Firstly, fillets were added to each corner to alleviate stress concentration points (simultaneously removing the point of singularity). Secondly, the ends of the bracket upon which it is fixed were slimmed down which did not significantly decrease the structural integrity of the bracket while decreasing weight. Lastly, cutouts in the form of circles were made into the material. The bracket can be thought of as a collection of cantilevered beams with normal force applied at the free ends. This force creates a bending stress within the material with the maximum condition occurring at the fixed support end.

Bending stress can be equated as follows: $\sigma = \frac{M*c}{I}$

Applying cutouts counterintuitively improves the structural integrity of the bracket by decreasing the cross sectional polar moment of inertia, I, of the bracket and thereby decreases the induced bending stress.

The final weight of the design was measured also using Abaqus mass properties: 16 kg. This is a 27% weight reduction as compared to the original design.



```
Volume centroid: 112.17,121.45,0
Mass: 1.60e+10
Center of mass: 112.17,121.45,0
```