DESIGN OPTIMIZATION OF RACE CAR STEERING KNUCKLE FOR ADDITIVE MANUFACTURING

by

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Abstract

DESIGN OPTIMIZATION OF RACE CAR STEERING KNUCKLE FOR ADDITIVE MANUFACTURING

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Additive manufacturing enables increased geometric complexity in structural configuration compared with conventional manufacturing methods. A Physics-first Computer Aided engineering (CAE) process beginning with structural topology optimization enables engineers to take advantage of this increased geometric design freedom. This work develops and demonstrates the steps and tools necessary to realize complex design configuration for additive manufactured. This report looks at the race car steering knuckle using the UTA FSAE upright as a case study. The stiffness driven upright component design is executed using topology optimization and NURBS based surface modeling tools. The topology optimization model is developed for 3 different load conditions driven by a maximum stiffness objective and volume fraction constraint. The resultant coarse, noisy, meshed isosurface is then translated to smooth Non Uniformed Rational Basis Spline (NURBS) based geometry. This work outlines the geometric operations and complexities involved in realizing the design. This process development effort shows the current capabilities and limitation of commercial Computer Aided Design (CAD) software and outlines procedures to realize complex design configurations for additive manufacturing.

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Chapter 1

Introduction

Additive manufacturing (AM) is a form of manufacturing that enables increased geometric complexity. AM, also known as 3D printing in the recent years, has grown and now has the capability of working with more material and different applications [1]. This advanced form of manufacturing is not only for prototyping but can be used to make customer end products. This ability to have complex geometry allows the designer to have higher freedom to model and design such geometry. In structure engineering this is one of the big constraints that has been removed from structural design compared to using conventional manufacturing [2]. Structure engineers can design structures with a physics-first design instead of a design for manufacturing (DFM) approach. Physics-first design focuses on structure properties and performance of the structure [3].

Computer-aided engineering (CAE) is used to help with the structure design and verification of the geometry. Also to be efficient with the process of design, optimization will be used. The design is optimized for performance with given conditions and goals. To illustrate the concept of a physics-first design, the University of Texas at Arlington (UTA) Formula Society of Automotive Engineers' (FSAE) upright will be used in this thesis [3]. The main goal is to lower compliance and if possible reduce the weight compared to the current design on the FSAE team upright [4].

This thesis will focus on topology optimization which is used to determine where material is needed in a given design space by following the load paths [5]. Before topology can be performed, the model needs to be understood and properly constructed to get accurate results from the optimization. The upright was applied under three main load cases: braking, turning left, and turning right. These load cases will be the

determining factors on how the upright performs. Also, the model needs to be constrained with supports to react the loads [6]. Next to get topology results the model needs to have specific goals, but in this thesis the main goal is to lower compliance. In this thesis the last thing needed to get results is to set limits on the model by applying both volume fraction and thickness of members.

Once the optimization results have been interpreted, a solid model needs to be made from this interpretation. Creating a solid model is a challenge and standard computer-aided design (CAD) software will not be good enough to create this solid model. In this thesis, to cope with the complexity of the optimization result, industrial modeling software will be used instead. The reason for using this industrial software is for the use of Non Uniformed Rational Basis Spline (NURBS) [7]. This modeling tool is the key to creating complex surface geometry which can then be converted to a solid model. With the solid models, structure analysis can be performed and is also needed to get the modeled manufacture with AM.

This paper is organized as follows. Chapter 2 discusses the background of structure optimization and needs information to understand topology optimization. This chapter will also cover the topic of NURBS, how they work and interact. Last this chapter will get technical information of AM and its contribution to complex geometry. Chapter 3 will cover the upright and optimization of the upright in detail as well as the process to perform topology optimization. Chapter 4 will focus on surface modeling and difficulties that comes with creating models for topology results. Chapter 5 will summarize the results and layout future work.

Chapter 2

Background

In the recent years there has been a big improvement in the field of additive manufacturing, structure optimization and surface modeling. It is important to understand each one of this subject, how they work, and when to apply them correctly. This section will discuss the technical details involved and effects they have on each other. This information sets the foundation for the rest of this thesis. In Figure 2-1 the design cycle is shown from my senior design [4].

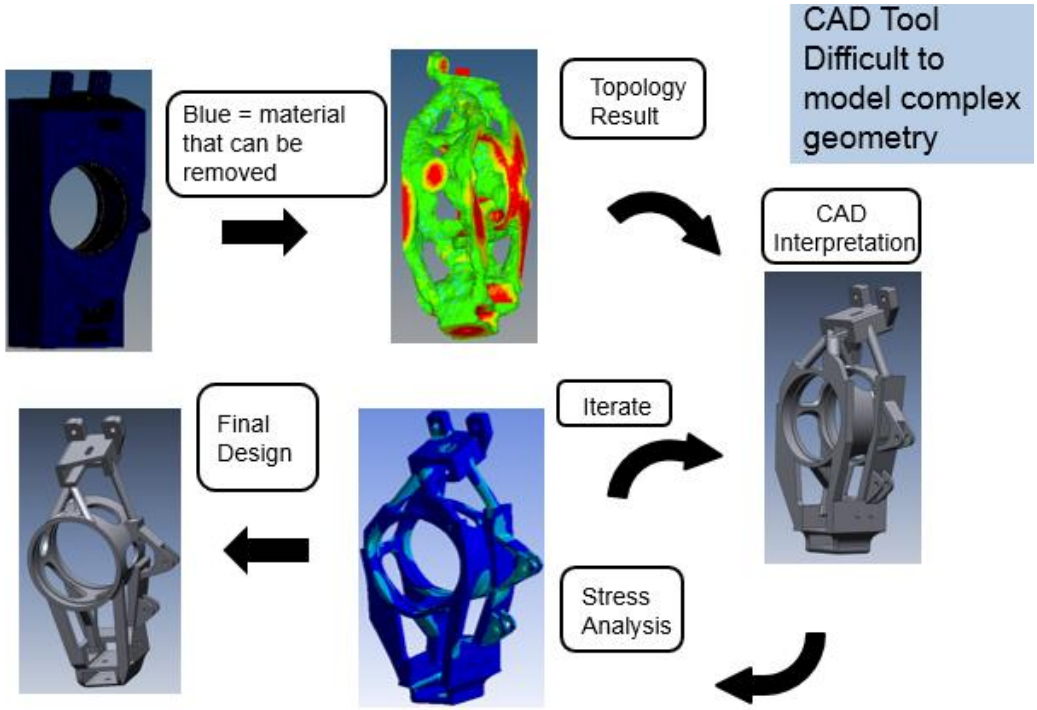


Figure 2-1 Optimization Cycle

2.1 Structure Optimization

Structure Optimization is a mathematical approach known as the material distribution method, spreading material in a layer in a given design space for a given set of loads and boundary conditions [5]. This set of optimizations is split into sizing, shape and topology optimization as can be seen in Figure 2-2. Structure optimization is nicely stated by N. G. Iyengar [8]:

> "Optimization techniques play an important role in structure design, the very purpose of which is to find the best solution from which a designer or a decision maker can derive a maximum benefit from the available resource".

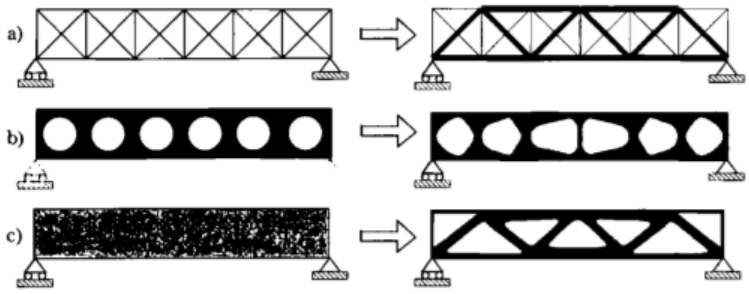


Figure 2-2: a) Sizing, b) Shape, c) Topology [5]

2.1.1 Sizing Optimization

Sizing Optimization is one of the three different types of structure optimization that are used to create a conceptual design. Sizing optimization defines ideal parameters to a component. The parameters are a set of constraints that the components must comply with while at the same time trying to achieve a set goal and or goals. These components are usually defined as a beam, shaft, and plates and so on. Some examples

of goals can be lower weight, lower compliance and reduction of cost. The main objective of the sizing optimization is to determine the ideal thickness of any of these components to achieve its goal or goals. The sizing optimization generally comes after shape and topology optimization because it needs initial geometry of the component to be defined before it can be run. Sizing is the last step in the structure optimization process.

2.1.2 Shape Optimization

Shape Optimization is one of the three different types of structure optimization that are used to create a conceptual design. Shape optimization defines ideal parameters to a section. The parameters are a set of constraints that the section must comply with while at the same time trying to achieve a set goal and or goals. These sections are usually defined as members, walls or shapes. Some examples of goals can be lower weight, lower compliance and reduction of cost. The main objective of the shape optimization is to determine the ideal shape of any of these sections to achieve its goal or goals. The shape optimization generally comes after topology optimization because it needs initial spread of the material to be defined in a design space before it can be run. Shape is the second step in the structure optimization process.

2.1.3 Topology Optimization

Topology Optimization is one of the three different types of structure optimization that are used to create a conceptual design. Topology optimization is a mathematical approach that will spread material where it is needed within a given space. This material is optimized when loads and boundary conditions are applied and design requirements

are stated. These design requirements can be set values of performance; for example, safety factor, weight or size. The approach is to minimize/ maximize a certain objective function f(x) [9] with all the set parameters. In 1988 Bendsoe and Kikuchi introduced a seminal paper on the methodology to homogenization for computing optimal structure designs [10]. Later in 1991, an alternative to topology optimization was introduced called solid isotropic material with penalization (SIMP) by Zhou and Rozvany [11]. SIMP is used to discretize the design domain. The variables are the elements which are penalized to some power times the material properties of the solid material. In 1997, a new method was introduced called evolutionary structural optimization(ESO) design methods by Xie and Steven [12]. ESO simplifies the math and reduces programming time needed to solve. It has also worked very well for compliance minimization cases. ESO takes its concept from nature and how bones, trees and shells have evolved over a period of time to get the optimal structure for their particular environments. ESO works in the same way. As it continues to iterate, it finds material that can be rejected based on rejection criteria such as stiffness, buckling and temperature. This method use finite element analysis (FEA) to get the information to find the rejectable material. In 2001, an educational report was written by Sigmund showing a compact Matlab topology optimization code to find minimal compliance in a loaded structure [13]. The Matlab code includes the optimizer, filter and finite element subroutine only using 99 lines of code. This Matlab code will be improved in 2011 by Signmund and Anderson by reducing its lines of code to 88 [14]. This new code is taking advantage of Matlab loop vectorization, memory preallocation and restructuring of the old code. These features reduced computation time and memory usage, and added density filtering.

It is also important to look at the math that is behind the topology optimization. In this paper, the modified SIMP method is used for solving topology optimization problems

following the simple beam problem used in the 88 line code [14]. The condition for this problem is the use of isotropic material, which mean the physical properties are the same in all directions. The topology optimization is using a derivative approach, which means that material is removed from the overall starting volume. Minimizing compliance is the goal for doing the optimization. It is density driven, which means that material is allocated where displacement is highest. The design region is discretized by a square finite element and density based approach, each element e is assigned a density x_e and Young's Modulus E_e:

$$E_e(x_e) = E_{min} + x_e^p(E_0 - E_{min}) \qquad x_e \in [0,1] \tag{1}$$

Where, E_0 is the stiffness of the material and E_{min} is a small stiffness assigned to the equation to deal with the void regions. This prevents the stiffness matrix from becoming singular. The p is the penalization factor for intermediate elements. In Figure 2-3, it can be seen how having a higher order of penalization factor increases the order of the ratio. Equation (2) uses the power law representation of elastic properties to show how penalized intermediate densities force final design to be represented by densities of 0 or 1 for each element [15]. The $\underline{K}$ and K represent the penalized and real stiffness matrix of an element while ρ is the density of the element. The number for p starts at 1 and can go higher depending on the design. For example, if there were a minimum member size, then the penalty would increase to 3.

$$\underline{K}(\rho) = \rho^P K \tag{2}$$

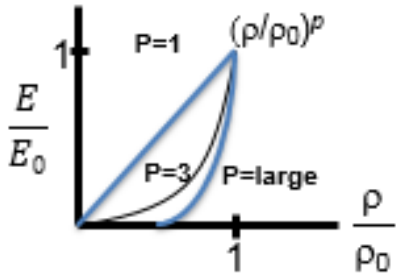


Figure 2-3: Penalization graph [3]

Equation (1) is known as the modified SIMP method. This differs from the original by having a E_{min}. In the original SIMP, elements with zero stiffness are avoided by imposing a limit slightly larger than 0 for the densities x_e. Equation (3) shows the mathematical formulation of the optimization problem:

$$\begin{aligned} \text{Min x:} \quad & \mathrm{c(x)} = U^T \mathrm{KU} = \sum_{e=1}^{N} E_e(x_e) u_e^T k_0 u_e \\ \text{Subject to:} \quad & \frac{V(x)}{V_0} = f \\ \text{Structure:} \quad & KU = F \\ \text{Density:} \quad & 0 \leq x \leq 1 \end{aligned} \tag{3}$$

Where c is the compliance which is what we want to minimize, U and F are the global displacement and force vectors, respectively, K is the global stiffness matrix, u_e is the element displacement vector, k_0 is the stiffness matrix for an element with unit Young's modulus, x_e is the vector of design variable (element densities), N is the number of elements used, p is the penalization power, $V(x)$ and V_0 are the material volume and given volume domain, respectively, and *f* is the volume fraction, also called the design constraint.

2.2 Modeling

Once a structure has been optimized, the next step is to create a solid model. Creating a solid model is usually done with a computer aided design(CAD) software and then exported to be manufactured [2]. Interpreting these topology results in CAD is very difficult because of its complexity, and it needs a different approach to create this solid model.

2.2.1 Alternative Modeling Techniques

One method to create this solid model is to export the triangulated mesh topology result as a cloud points, and then use that format to create a surface model with mesh patches as introduced by Kobbelt and Botsch [16]. The issue with cloud points is that with topology optimization, the number of points can be very large and can make poly meshing very difficult as can be seen in Figure 2-4. In one report by Fabio [17] he goes through many different techniques to try to deal with the issues that come from point clouds, but even then it is very time consuming for the engineer, uses many different software programs and has a high computation time.

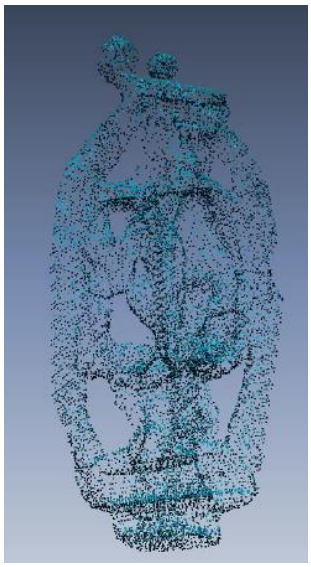


Figure 2-4: Point Cloud of Upright [4]

Last note on cloud points to surfaces comes in the idea of reconstruction by using filtering, mesh optimization and reducing the noise to create a meshed surface as written by Tishchenko [18]. The issue with using this method is it reduces the resolution of the topology results and can lead to mesh errors.

Another approach is to model the part the same way it is going to be manufactured. In the case of additive manufacturing, it is made by layers as presented by Ahasan and Habib [19]. The process is to parallel slice the model in the build direction, then sketch the outline of the topology results in each layer as seen in Figure 2-5. This modeling is similar to lofting with the added benefit of having disjointed sketches in one plane. The issue with this method is that it requires many planes, sketches in each plane to get an accurate representation of the topology results and is very poor at bending different members together.

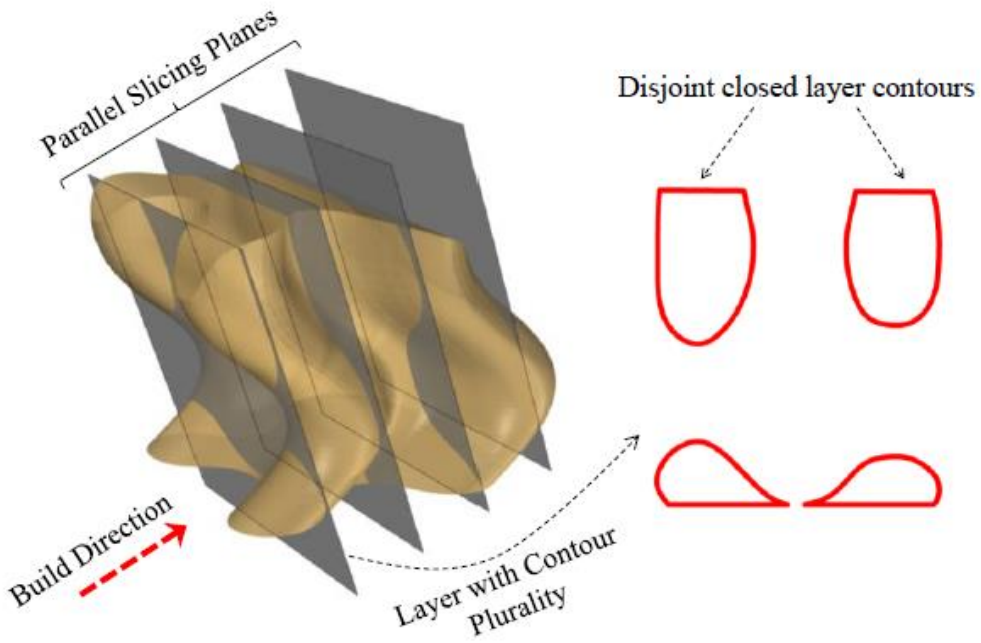


Figure 2-5: Slice Modeling [19]

2.2.2 NURBS

Non-uniform rational B-spline (NURBS) is a common modeling tool used for generating curves and surfaces [20]. NURBS is also widely supported by many CAD programs and works well with IGES files. The great benefit of NURBS is its stability, flexibility, ease of implementation and robustness to noise [7]. The down side of NURBS is its interaction with other CAD tools are not continuous and maintenance can be very time consuming. In the work by Leal and Branch [7] they show an automatic, efficient and simple method for constructing NURBS surfaces from point clouds. The NURBS are defined by equation (4). The w_i is the weight of the points, P_i are the control points and $N_{i,p}(u)$ is the nonrational B-spline basis function defined by vectors [21].

NURBS Curve

$$C(u) = \frac{\sum_{i=0}^{n} N_{i,p}(u) w_i P_i}{\sum_{i=0}^{n} N_{i,p}(u) w_i} \quad (4)$$

$$0 \leq u \leq 1$$

Equation (5) is for NURBS surfaces, the $w_{i,j}$ is the weight of the points, $P_{i,j}$ are the control points and $N_{i,p}(u)$ and $N_{j,q}(v)$ are the nonrational B-spline basis function defined by vectors in their respective plane [21].

NURBS Surface

$$S(u,v) = \frac{\sum_{i=0}^{n} \sum_{j=0}^{m} N_{i,p}(u)\, N_{j,q}(v) w_{i,j} P_{i,j}}{\sum_{i=0}^{n} \sum_{j=0}^{m} N_{i,p}(u)\, N_{j,q}(v) w_{i,j}} \quad (5)$$

$$0 \leq u, v \leq 1$$

Control points are what define the line and are also known as the local support property, and this means that the surface can be affected only locally. This is very helpful because when one point is modified, the rest of the point and their properties stay the same. The weight factor is what gives the control points the influence to shape the curve and surface and can be seen in Figure 2-6.

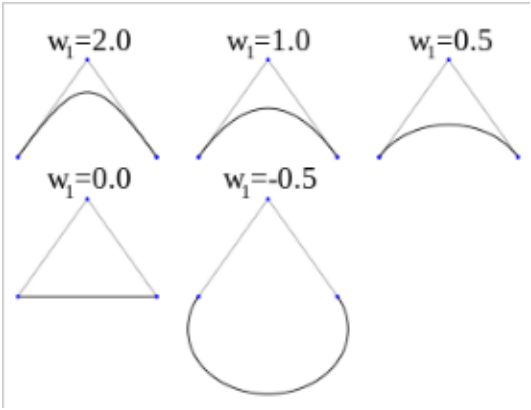


Figure 2-6: Different Weight Factor [20]

2.3 Additive Manufacturing

AM is formally known as rapid prototyping (RP) but as technology has grown the idea of AM has changed as well. RP is described as a process to quickly create something to physically show the design before its final commercial version [1]. AM is now capable of creating commercial ready parts, and calling it prototyping is no longer correct. AM, also known as 3D printing, works by making a part layer upon layer, with each layer having a cross section derived from the CAD model. In AM, the more layers the more detailed the part will be. This difference in AM will determine the mechanical properties, surfaces, material, and limitation on the models. Most AM still follow the steps in Table 2-1

Table 2-1: AM Guide Lines

1	Create CAD model
2	Convert CAD to .STL
3	.STL file manipulation
4	Machine Setup and Slicing
5	Physical Built
6	Remove Material
7	Post-Processing
8	Application

AM also has the important feature to allow the engineer the freedom to design models with a physics approach and not a manufacturing design approach. AM is a form of manufacturing and with it there are many different systems with benefit and special application. In this paper the formula upright is looked at as case study to show AM and optimization working together. This part needs to be very stiff but have some ductility so metals are a good candidate. The part also needs to be light on weight so aluminum is chosen for its high strength-to-weight ratio. Lastly, the part is optimized and will have complex geometry. The AM system will need very good resolution and good mechanical properties.

2.3.1 EBM

Electron beam melting (EBM) is a laser based AM process that can work with metals. EBM uses high-energy electron beam to induce fusion between metal powder. This beam is controlled by deflection coils, Figure 2-7. The base plate lowers layer by layer. With each layer a new set of metal powder is overlaid by the powder scraper. EBM can create very dense parts and can work with a wide range of metals like aluminum and titanium.

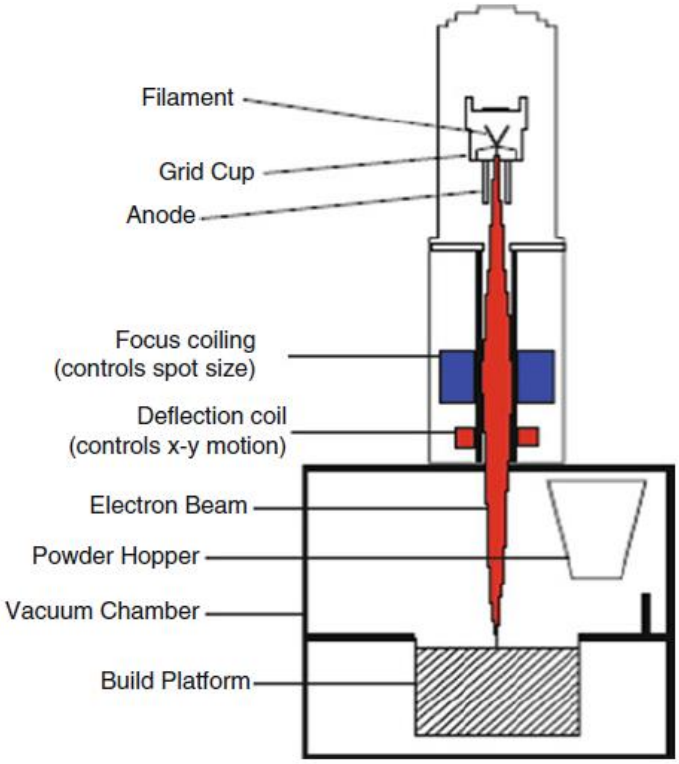


Figure 2-7: EBM process [1]

2.3.2 SLM

Selective laser melting (SLM) is a system that uses a laser based system for direct melting of the metal powders. SLM is ideal for the manufacturing of the upright. Figure 2-8 shows the SLM process. The laser hits a set of mirrors which are then deflected, which controls the contour of the laser. The base plate lowers layer by layer. With each layer, a new set of metal powder is overlaid by the powder scraper.

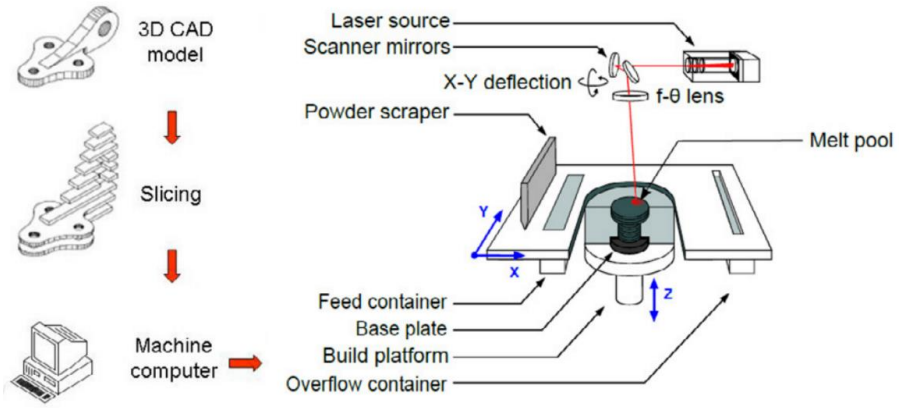


Figure 2-8: SLM process [22]

In Table 2-2 the pros and cons of SML compared to EBM can be seen.

Table 2-2: SLM Pros & Cons compared to EBM

SLM-Pros	SLM-Cons
Fiber Laser Based	Not as Dense as EBM
Excellent Surface	High Energy Usage
Highly Complex parts	Preheating requires a separate heat source
High Resolution	Support needed for thermal stresses
Low preheating Temperature	Hard to remove Support Material
Wide Material Selection	Controlled print environment of inert gases

Chapter 3

Front Upright Structure Design Optimization

This section covers the model in detail, topology optimization and the results of the optimization. It also takes a look at the INSPIRE the topology software and how to properly apply loads and supports. The upright is made of aluminum and is manufactured through computer numeric control (CNC) machines [23]. The main goal of the optimization is to minimize compliance compared to the original upright without gaining too much weight. The upright was a good component to use because it is a structure part that could reduce weight and also show that AM can be applied to complex parts.

3.1 Front Upright model detail

The front upright used in this paper is the one used from the University of Texas at Arlington (UTA) Formula Society of Automotive Engineer (FSAE), Figure 3-1 shows the front upright inside the wheel. The upright is the part that connects the control arms to the hub; the hub then connects to the wheel [24]. The upright also connect the steering arm to turn the car and brake calipers to stop the vehicle. The steering arm and break caliper are bolted to the upright and the control arm connects to the upright with a ball joint. The hub connects to the upright through bearings which allow the hub to rotate as the upright remains stationary relative to the chassis. The upright needs to hold all these components and remain stable.

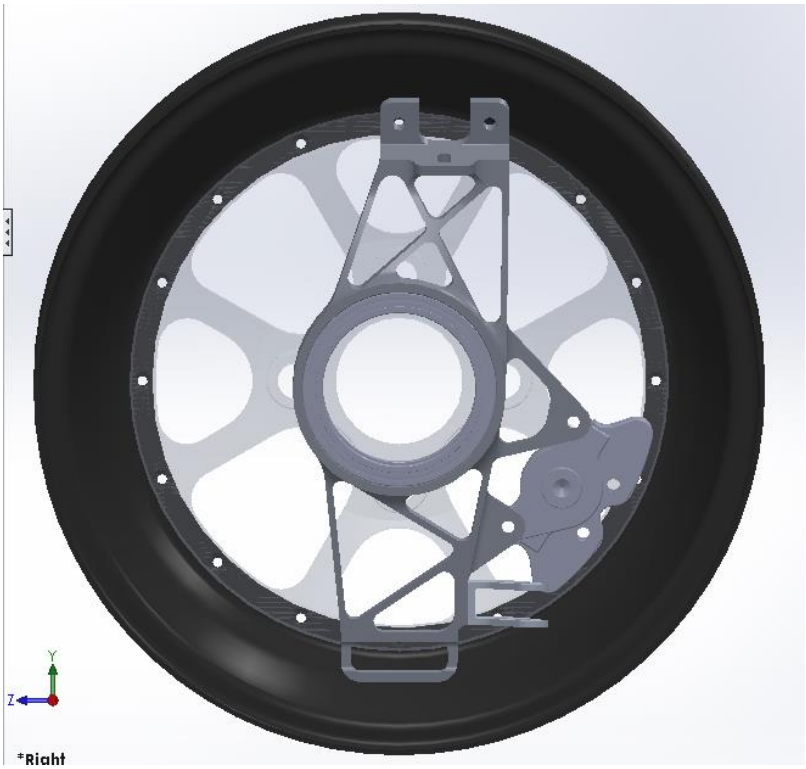


Figure 3-1: Upright inside Wheel

It is also important to understand the loads that are applied on the upright and the different load cases. In Figure 3-2 the free body diagram of the wheel assembly is seen for the load case for breaking. The vertical load is half the weight of the car for the weight is split between both sides plus the load transfer from the car breaking. The longitudinal load is the force from the tire and the ground as the car is breaking. The load by the calipers is very high, to take a worst case scenario, when going at 2gs and breaking. The next scenario is turning and in Figure 3-3 shows the free body diagram of the wheel assembly when turning. The lateral load is from the friction of wheel to ground and the vertical load is from the weight of the car. This load case also has the steering load created by the steering arm.

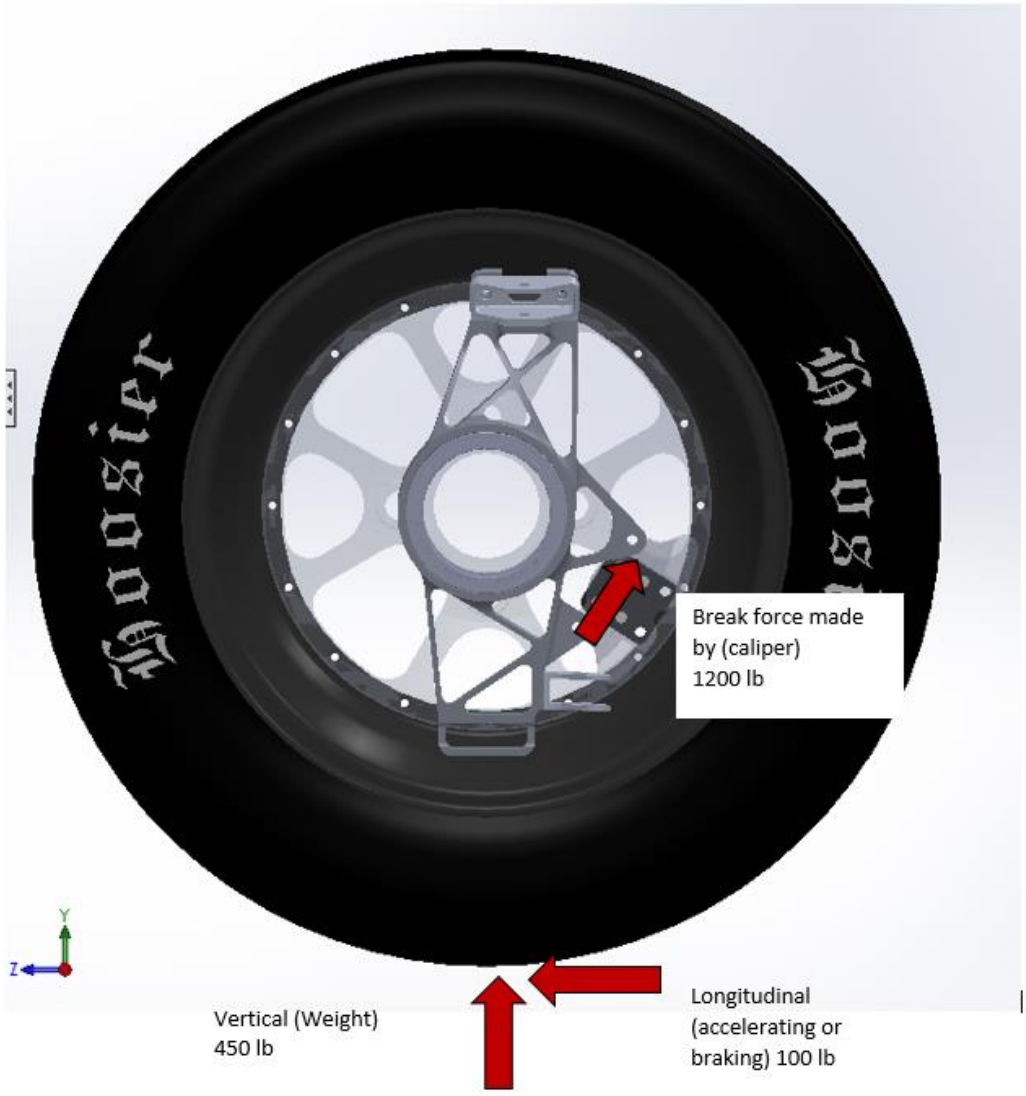


Figure 3-2: Free body diagram of front right wheel assembly braking

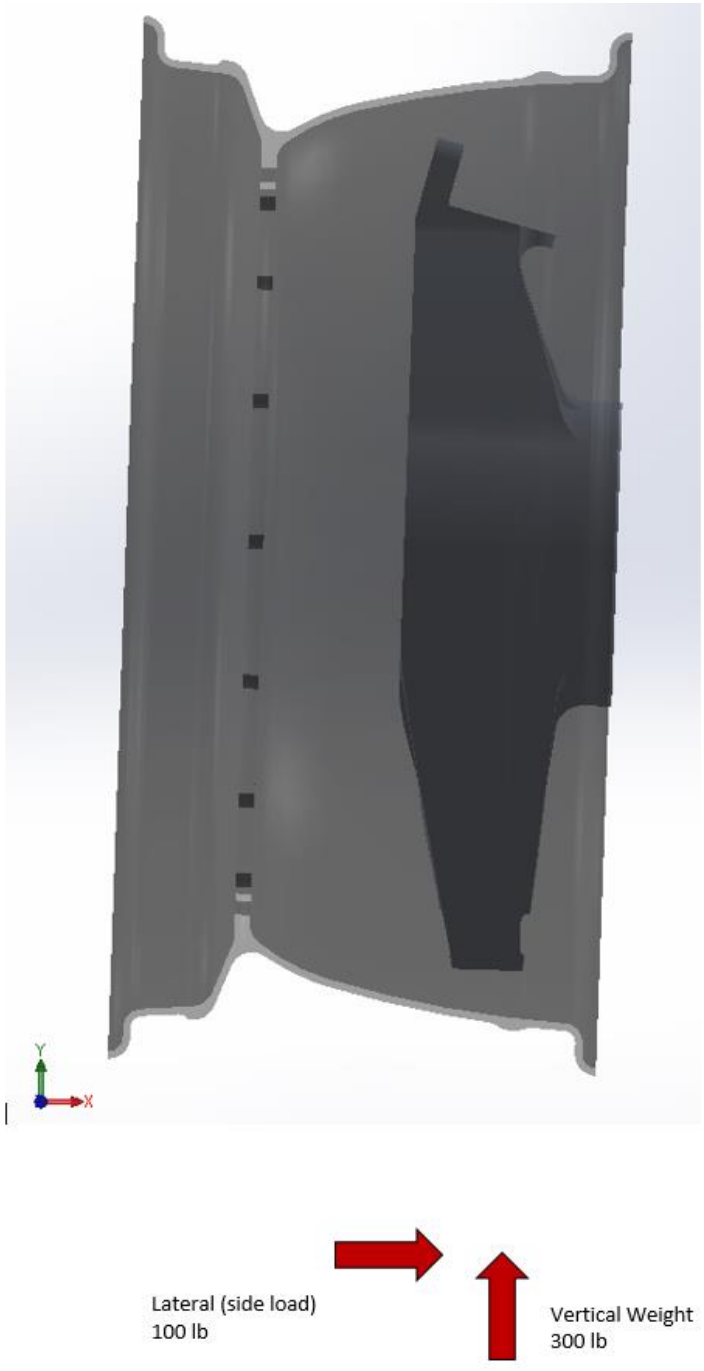
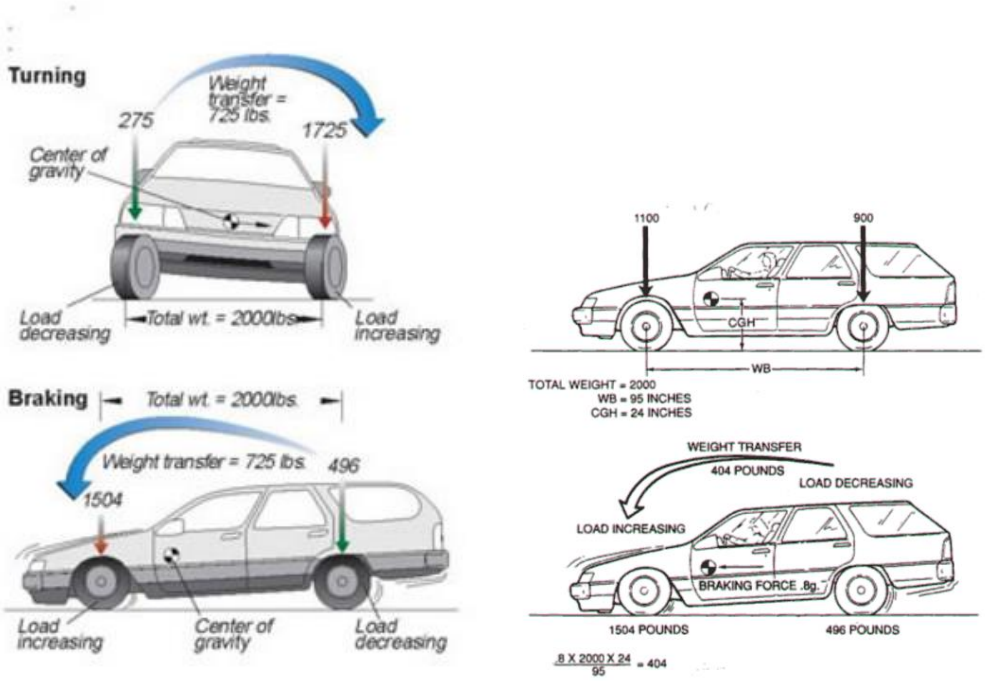


Figure 3-3: Free body diagram of front right wheel assembly turning

The load case of turning left is felt differently on the front right upright compared to if it were turning right because of the load transfer. Figure 3-4 shows the weight transfer, and is important to create another load case for turning in the other direction. Figure 3-4 also shows why the vertical load in the breaking scenario is higher than in the turning scenario.



(a) (b)

Figure 3-4: Weight Transfer (a) Turning and Braking and (b) Braking [25] & [26]

In the case of optimization, it is good practice to transfer all the loads and supports to the actual part that is going to be optimized. Topology optimization runs on FEA, so the less elements (less parts) the faster the results. In Figure 3-5, the loads from the wheel assembly have been transferred on to the upright itself. The cylinder in blue represents the support regions, the vertical load is reacted in the center of the wheel, the longitudinal load creates a moment and the braking load is coupled and separated.

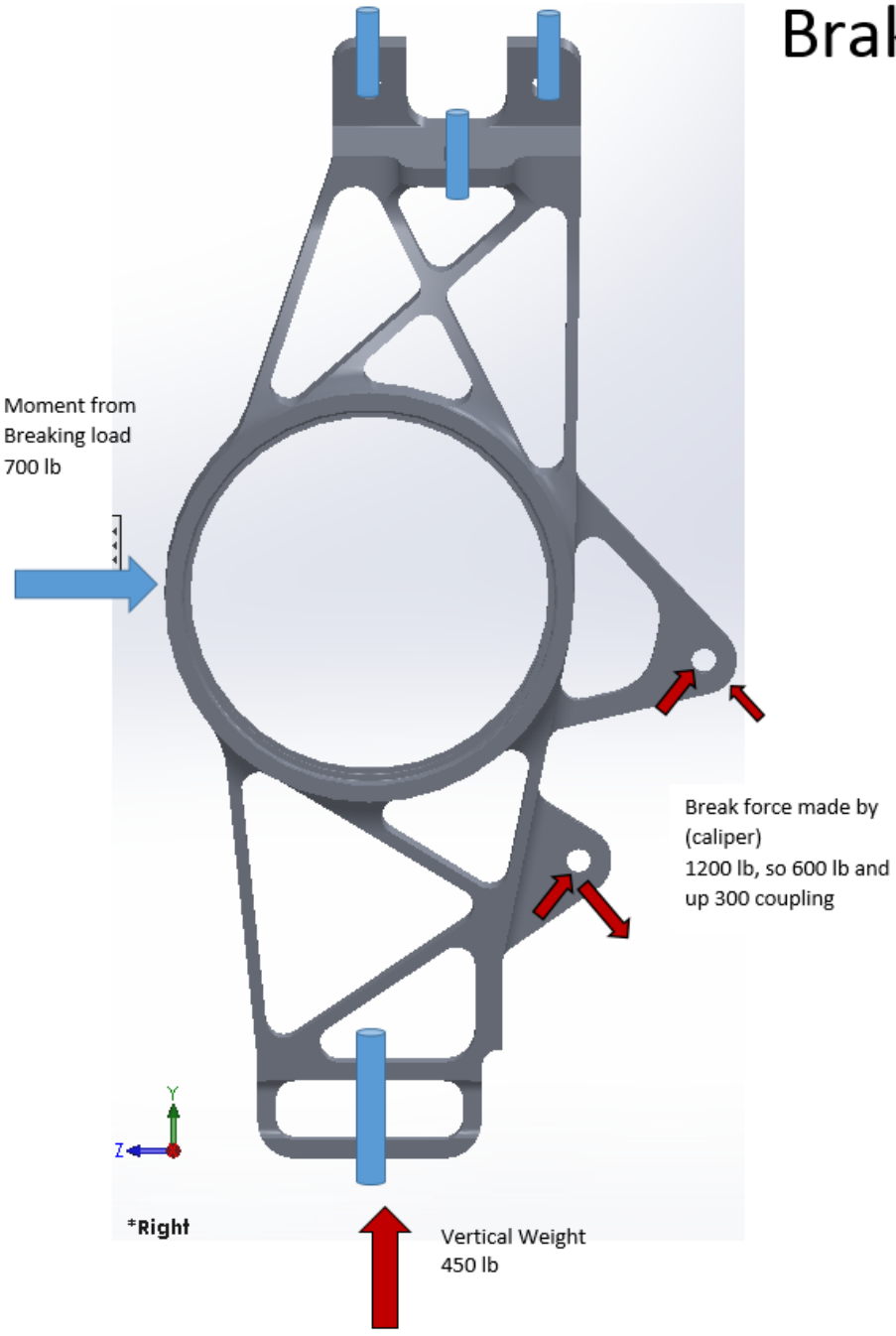


Figure 3-5: Free body diagram of front right upright braking

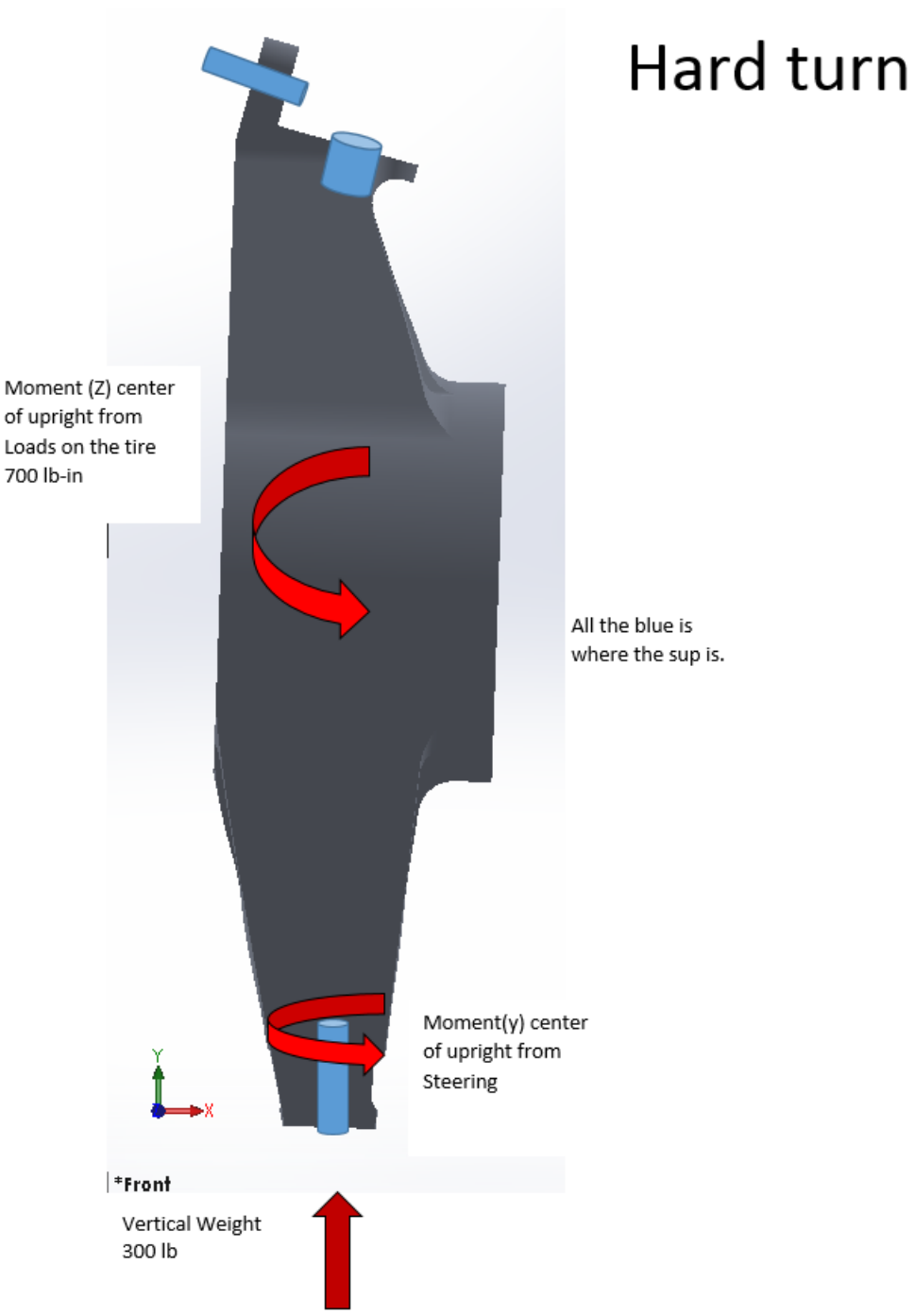


Figure 3-6: Free body diagram of front right upright turning

The free body diagram in Figure 3-6 is for the condition of hard turning. The blue cylinders represents supports and the vertical load is reacted on the center. The moment in the Z axis is created from lateral loads, and a moment in Y axis is also created from the steering arm loads.

3.2 Upright optimizations

The upright is optimized using Altair's SolidThinking INSPIRE. The objective of the optimization is to the highest stiffness with the given applied loads and boundary conditions. INSPIRE is owned by Altair and uses OptiStructs solver to get optimization results [27]. The OptiStruct solves the problem by using design variable, response, design constraint and objective function (DRCO) [15]. The reason for using INSPIRE over OptiStruct is for the simplicity that comes with INSPIRE. This software was made to take out the FEA from the user and just have it run in the background. Taking out the FEA makes it easier and faster to get results. This also allows designers who are not structure engineers to do topology optimization. This can lead to some drawback since the user has no control of the FEA. The results are constrained to the setting that INSPIRE has defined for the FEA, for example, mesh control. Another limitation is how loads and other conditions are applied, such as remote force and moments. INSPIRE also has the benefit of working with its brother software EVOLVE, which is an industrial modeling software. EVOLVE is important since it has the capability to create complex surfaces from the complex results that come from topology optimization.

3.2.1 Apply conditions and constraints

Starting the optimization there needs to be a CAD model that can be imported into INSPIRE. Next the model needs to be separated into different non-design areas.

These areas are where the model cannot change because they are critical for the performance of the model and its interaction with its surroundings. In the case of the upright, Figure 3-7 shows the grey area to be design space (material that can change) and the green is material that cannot change (non-designs area). These non-design areas will also be where loads and supports will be applied. Since loads are applied to these non-design areas, it is important to create them correctly.

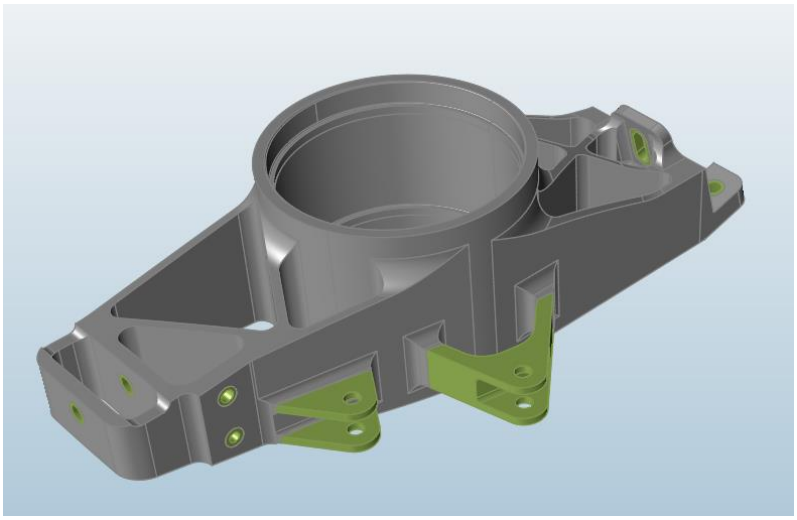


Figure 3-7: Non-Design Area

In Figure 3-8, it can be seen that a big ring has been created in the center of the upright, and this is to get a representation of the bearing that mates to the hub. This does two things: one, it makes it easier for the engineer to apply the moment at the center of the upright instead of coupling all the lateral forces, and allows ease of geometry manipulation. It is good to have all these parts named and organized to make them easier to work with later.

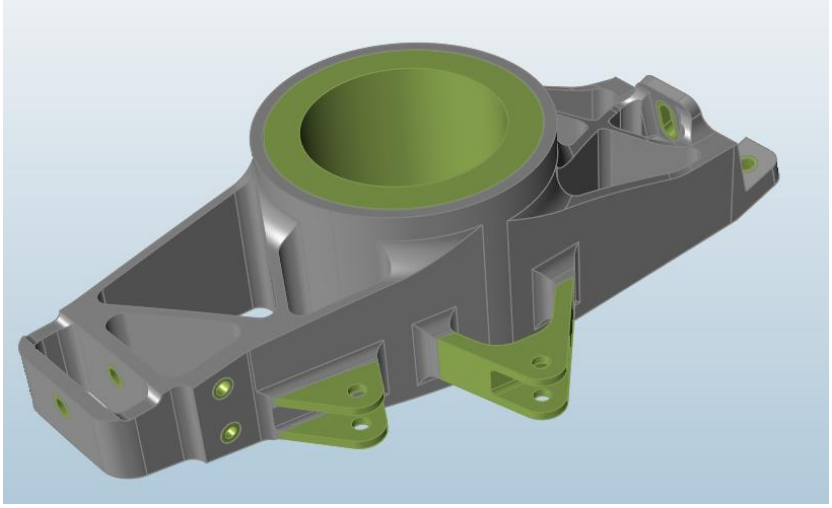


Figure 3-8: Non Design Area bearing

The next step is to define the package space, which represents the maximum volume that a part can occupy in its given space. It can allow the optimization the freedom to extend material to different locations. The upright fits inside a wheel, and that defines the dimension of the package space as can be seen in Figure 3-9. It is also important to have any cut out in the package space to allow for assembly, nuts, bearing and any other part that may need the free space to mate with the optimized model. In the package space, it is important to then define this volume as the design space as seen in Figure 3-10. Design space is the single solid shape that will be optimized and can be changed in any form. It is best to make the design space a simple shape, for example, a cube. The more detailed the design space, the longer it will take to get optimization results [27].

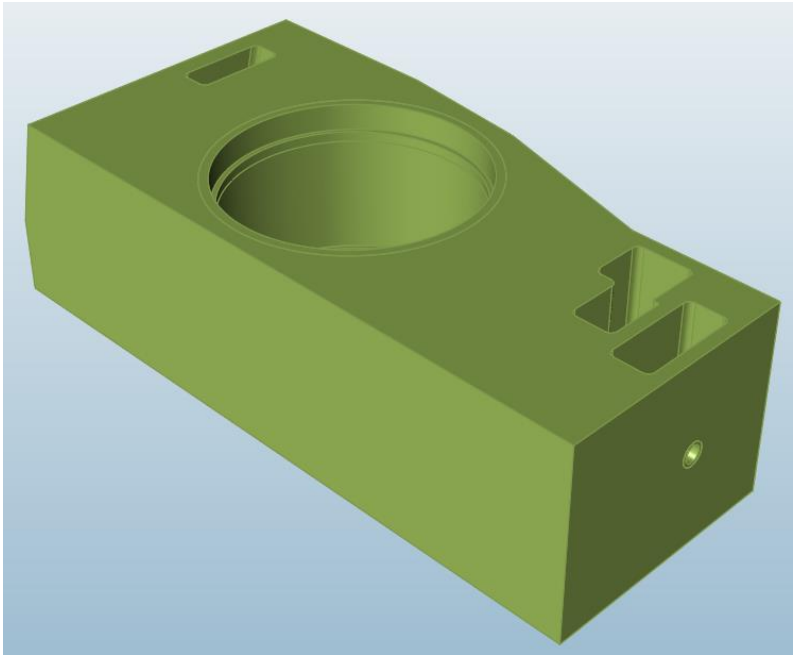


Figure 3-9: Packaged space

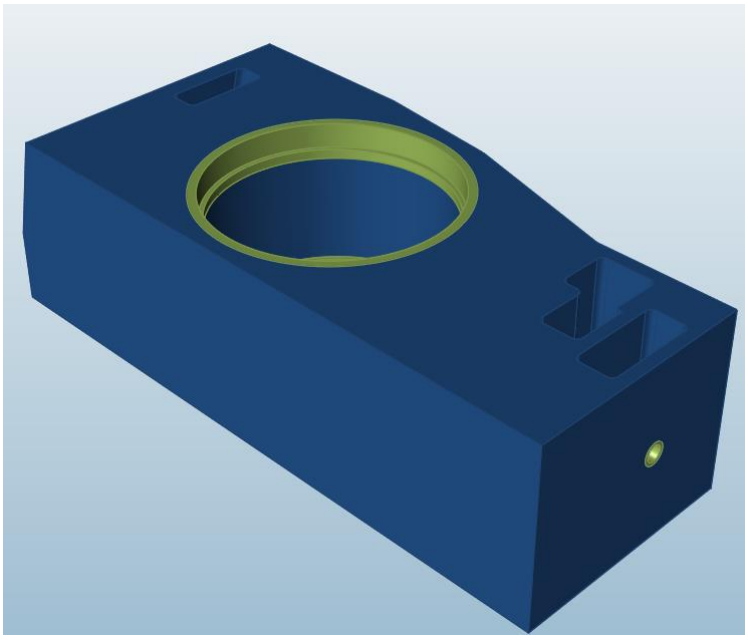


Figure 3-10: Design space

The loads seen in Figure 3-2 and Figure 3-3 are applied to non-design spaces. Each load case is applied to its non-design area. The one load that is special is the moment that is made at the center of the upright. Since all the loads are in the center, it would be ideal to create a surface that would represent the bearing. The bearing would transfer the load on to the upright as seen in Figure 3-11. All load cases must be applied to the upright which are hard braking and hard turning right and left.

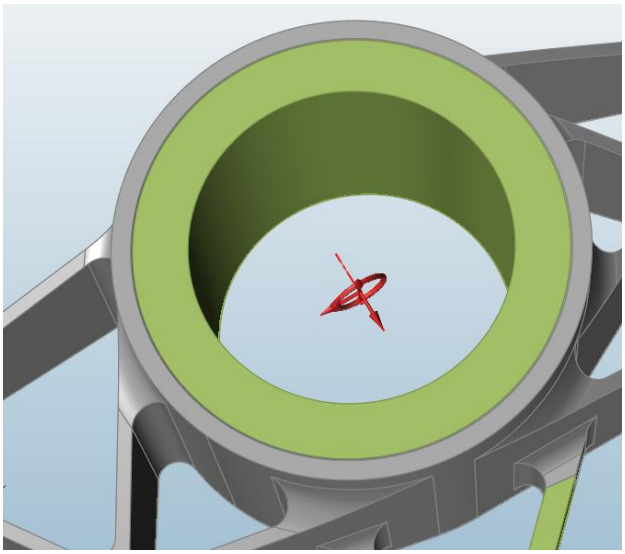


Figure 3-11: Center of upright

Applying support is a little different because this will be always true in all load case. When it comes to creating load cases it is ideal to have supports apply to all the cases and as seen in Figure 3-2 and Figure 3-3 the supports are the same. Another thing to look out for is the type of support, for example, fixed support or simply supported. On the upright the control arm mates to the upright with a ball joint, which is simply supported. Figure 3-12 shows the model can still rotate on that axis to represent the real world turning of the upright but still be stationary.

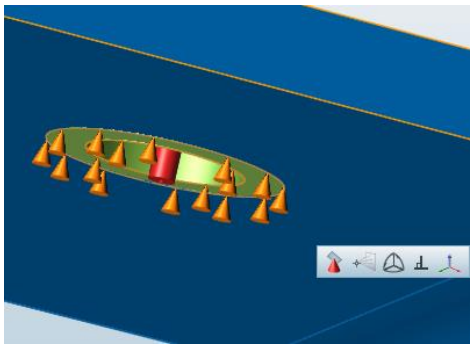


Figure 3-12: Upright Support

The last thing that needs to be defined is the material and contacts. In this paper only isotropic material properties which means that all the parts (design and non-design) will be the same. When it comes to contact regions, all the non-design areas came from the original solid model and are still a part of the single solid model which makes the contact bonded. In the case of the bearing a non-design area was created. The non-design will have some slipping as the upright turns so its contact will be separable.

3.2.2 Optimization parameters

Then comes the running optimization in INSPIRE. INSPIRE can perform either topology or topography optimization. Topology optimization follows the critical load paths to provide the optimal material distribution. Parameters are shown in Figure 3-13 starting with the objective which is the goal of the optimization, in INSPIRE the objective can be either maximize stiffness or minimize mass [28]. In this topology optimization the objective is to maximize stiffness with the given three load cases. The constraint for the optimization are the mass target and the thickness of the members. The mass target sets the volume fraction and it refers to the percentage of the initial design space that is to be

used in the final solution for the objective of maximizing stiffness. Thickness constraints can control wall thickness and the diameters of beam-like members in shape or close to by specifying a minimum and/or maximum thickness [28]. When it comes to the upright and the bearings there is separation between them. Selecting sliding with separation will produce more accurate results but the time to solve will increase greatly. When separation is selected the computation time went from 30 min to 2 days. The last parameter is the applied load cases on the optimization tool. To get the worst scenario all load cases are applied.

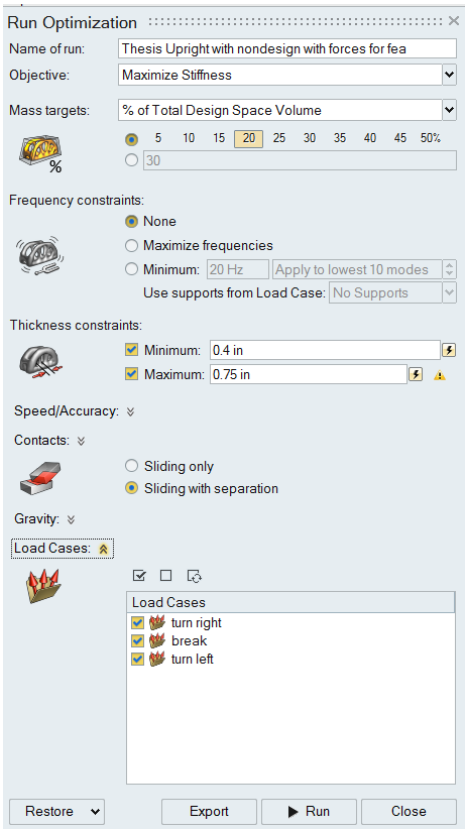


Figure 3-13: Optimization Run

3.2.3 Optimization Results

Running different percentages can change the results of the optimization greatly and its recommended that different runs be done. The different runs will help looking at design ideas as well as understand how the material is forming and the impact of the load paths. It is best practice to change one constraint at a time and not all of them at once because changing one at a time will help with understanding the impact each constraint has on the optimization. In Figure 3-14 the only constraint that is changed is the max value thickness. Material is now distributed out more and also making connection between non-design spaces. In Figure 3-15 the only constraint that is changed is the volume fraction. Material is increased and makes more solid and thicker connection as volume fraction increases. In Figure 3-16 the only constraint that is changed is the contact with separation and this has a big impact on the distribution of material. In the contact with separation the material needs to cover more area around the bearing to deal with the bearing sliding. After many runs and looking at the different results Figure 3-16 has the constraint of 20% volume fraction, .03 inches minimum thickness, .75 inches maximum thickness, and contact with separation. These results are very complex because of the nature of topology optimization and all the load cases involved. Another thing to note is that the results are not smooth. The rough and noise results makes it difficult to use any auto meshing software to create a solid model. The last problem with the results is that of some of the features are not complete and will require some interpretation from the engineer to create the solid model. The results can be saved as a .stl file and can be exported to any CAD or industrial free form software.

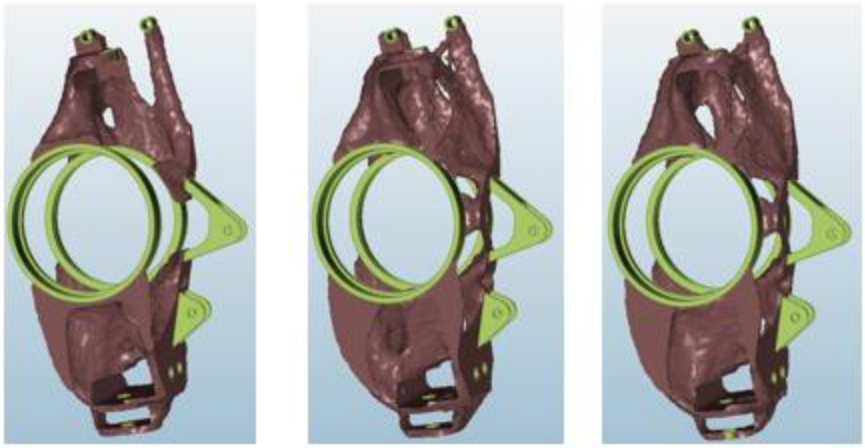


Figure 3-14: Max Thickness

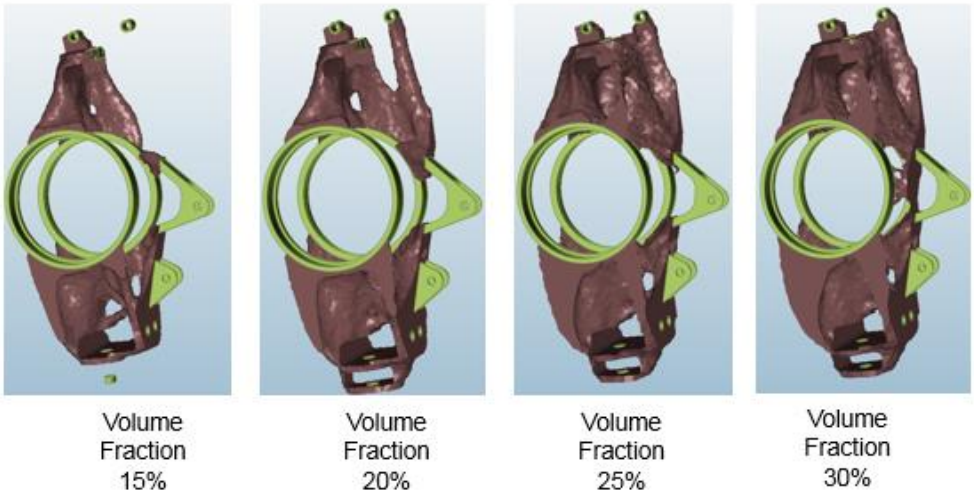


Figure 3-15: Volume Fraction

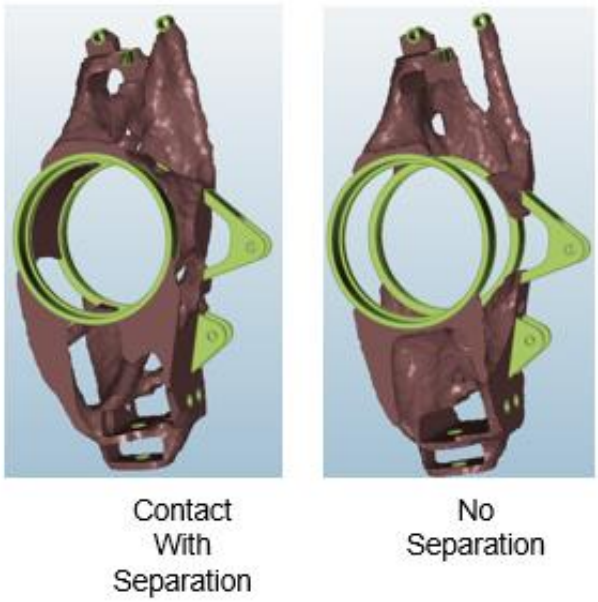


Figure 3-16: Contacts

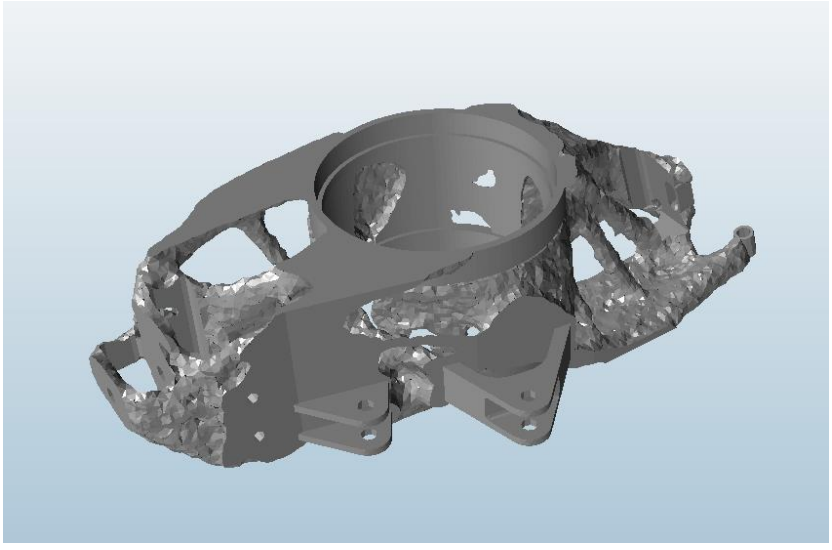


Figure 3-17: Topology optimization results – 20% volume fraction with thickness of min-0.3 & max- .75 inches and contact with separation

Chapter 4

Modeling

Solid modeling is a consistent set of mathematical and computer modeling of three-dimensional solids. Solid models have with them the visualization, animation, interior and consistency for a digital model to represent a physical object [29]. When it comes to doing engineering work, solid models are needed for calculating weight, FEA and physical machining application like .gcodes [23]. Solid modeling is great for simple geometry but as soon as the complexity is increased the solid modeling tools are not enough to create the desired model.

4.1 Freeform modeling

Freeform modeling also known as surface modeling is a technique to create a complex surface by the use of curves (splines) or blended meshes (lofted) [30]. This first method to surface modeling is good for continuous curved surfaces but not enough for organic surfaces. The second method is by directly creating surfaces by using surface poles/control points. This level of control gives the designer the freedom to create any shape. Freeform modeling is mainly used in computer aided industrial design (CAID) software. CAID is far more conceptual, creative, and less technical then CAD with all the physical properties [31]. This style of modeling is perfect for modeling the organic results from the topology optimization.

4.2 Surface Tools

The optimized upright is modeled using SolidThinking EVOLVE 2016. EVOLVE is owned by Altair which is one of the two programs created by SolidThinking. This CIAD software provides organic surface modeling and parametric control with NURBS based surfaces. Another feature from EVOLVE is that it has the same unique construction tree like that of a CAD software [32]. Other modeling tools that EVOLVE has are polygonal design, point clouds, automatic construction tree rebuilds, design iteration, and polyNURBS. When the topology results are imported to EVOLVE, it is best to just have the topology result imported as seen in Figure 4-1.

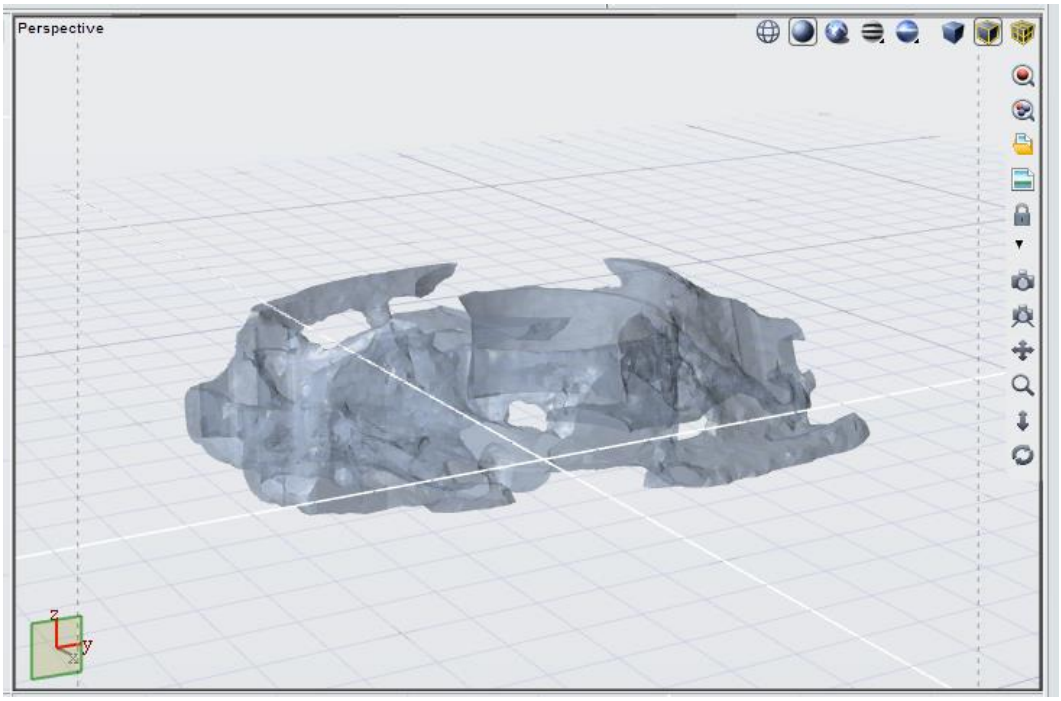


Figure 4-1: Imported Optimization Results only

4.3 PolyNURBS

To create the model EVOLVE 2016 PolyNURBS tool will be used in this chapter. The tools are displayed on the modeling left hand on the EVOLVE interface, the tools available are as seen in Figure 4-2. PolyNURBS is a modeling method that allows you to trace over the triangulated polymesh optimization results. It also creates watertight NURBS that can be converted to .stl files for AM [33]. PolyNURBS will start the user with a polygonal square and then be converted into NURBS as can be seen in Figure 4-3.

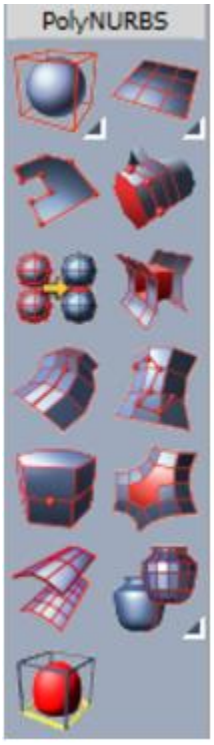
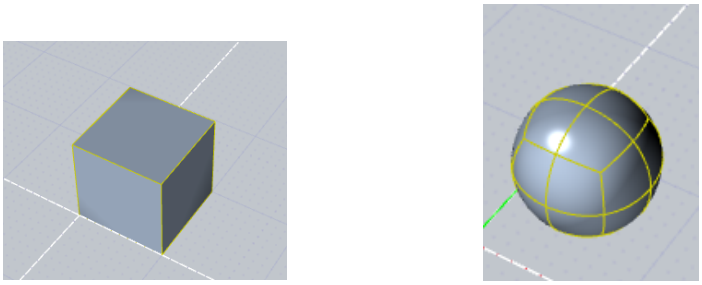


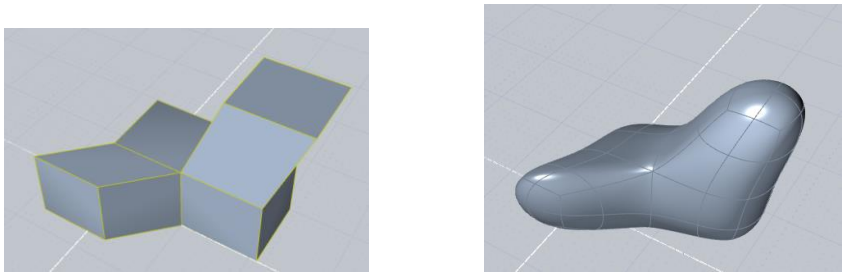
Figure 4-2: PolyNURBS Tool



(a) (b)

Figure 4-3: (a) Polygonal (poly cube) and (b) PolyNURBS

The polygonal can be controlled by changing a face, edge, and point. The control features allows the point to be rotated, translated, and scaled. Faces can be added and cut lines can be created to grow the polygonal. Nurbify the polygonal to get the smooth surface as seen in Figure 4-4. These faces, edges, and points can be deleted if needed but it is advised to never delete any of these faces ever. Another issue that needs to be avoided is to not have any of the blocks inverted. Inverting will cause the algorithm to run into problems and not create the polygonal.



(a) (b)

Figure 4-4: Extruded (a) Polygonal and (b) PolyNURBS (Nurbify)

4.3.1 PolyNURBS Interaction

In EVOLVE the topology optimization and the non-design space needs to be saved separately. The topology optimization can be traced over but cannot overlap the

non-design. It is also recommended that each created feature be in its own folder. Creating a new folder also keeps record of the overall model features. Another good practice is to lock the topology results. In Figure 4-5, the optimization folder is locked and everything inside that folder cannot be modified. Another good practice is to superimpose the polygonal over the imported noise topology results. Superimposing allows the engineer to see how close the model is to the results in real time. When overlapping the polygonal over the optimization results the optimization is not clicked by accident. Locking the folder will be very helpful later when the model needs trimming and rounding.

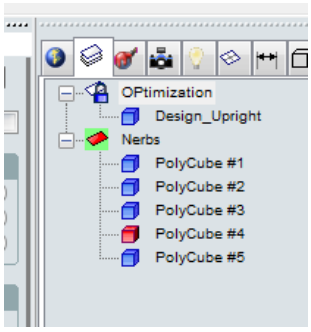


Figure 4-5: Organized Layer

4.3.2 PolyNURBS Sectioning

With complex geometry it can be helpful to split a section of the model into different regions. This helps with focusing on one region and then later combining all these different regions into one big polygonal. Creating polycube (blocks) to each feature is also recommended that way each feature, for example wall, member, or shaft, can be focused and then blended together. These recommendations can be seen in Figure 4-6.

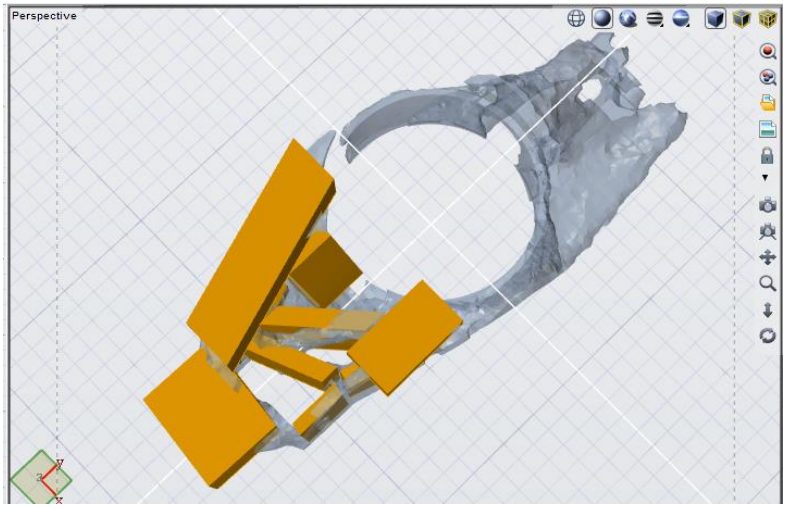
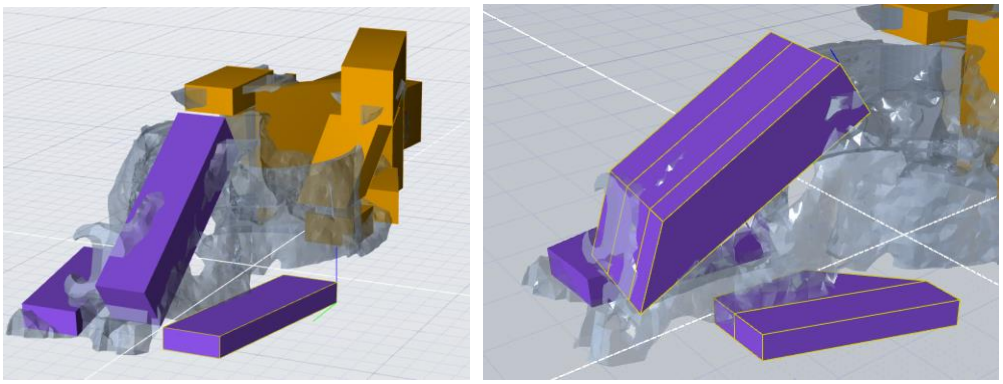


Figure 4-6: Low Region with polycubes cover the main features

The same work can be done to the other regions. It is helpful to lock the other regions that way there is no accident when clicking the different polycubes. It might be difficult to interpret each feature as just a block. To help with this a block can be cut and modified to match the optimization as shown in Figure 4-8. It is good practice to make cuts on a block so it has odd sections. In Figure 4-8 (b) the middle block is cut into three sections. This will be helpful later when blending to other blocks together. The section tool can be seen in Figure 4-7.

Figure 4-7: Section tool



(a) (b)

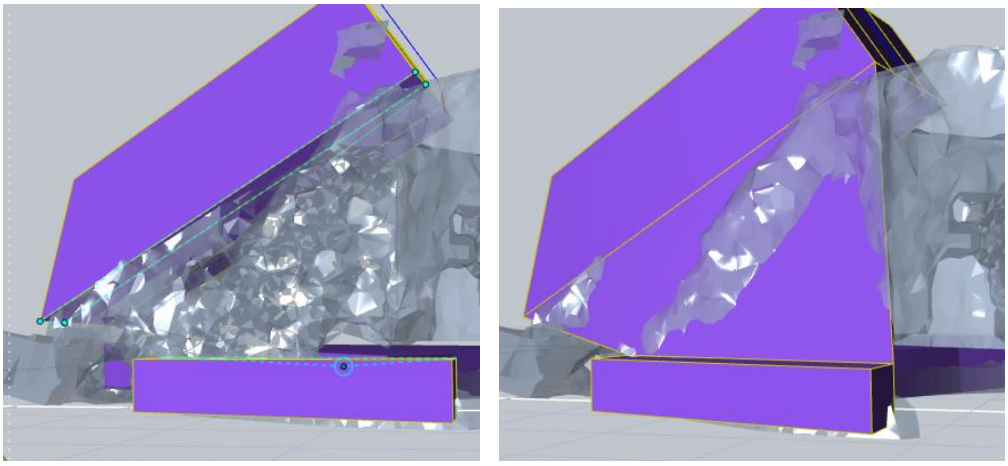
Figure 4-8: (a) Block and (b) cut & modified block

4.3.3 PolyNURBS Bridging

Once all the major features have been made into blocks and adjusted it is time to combine them. The combining from block to block is called bridging. Bridging is a tool that creates adjacent faces where bridging occurred but removes the internal faces of the PolyNURBS [34]. Bringing the blocks together can be seen in Figure 4-10. Before the blocks are bridged the must be combined using the tool in Figure 4-9 (a). Once combined the two block will be treated as one block but with a gap. After combining the blocks comes the bridging. In the bridging it is important to match the desired points to each other, this point indicate the mating edges. The reason for having odd number of section is if any modification need to be made to the bridge later, the rest of the block it is unaffected by this changes. Another important thing to keep in mind is to always keep the bridge faces the same as the number of edges. This will not be a problem as long as features don't get deleted or the block become inverted. This process of combining and bridging is repeated for the rest of the blocks and all the different type of blends in the rest of this paper.

(a) (b)

Figure 4-9: (a) Combine Tool and (b) Bridge Tool



(a) (b)

Figure 4-10: (a) Bridge select and (b) Bridged block

4.3.4 PolyNURBS Multi Blend

Bridging is very helping for connecting two different blocks together but in other situation a multi blend is needed. Multi blend is when three or more blocks blend together at one point. These blends can become very complex especially with the more blocks involved. Organizing such blends in 3D space can add to the challenge. This situation can be seen in Figure 4-11 red outline were the four blocks will need to come together. The best approach to multi blend is to focus on a one section at a time. Focusing will require experience of the user to predict how the final blend will look such as a painter

does with his art work. In this case of multi blends the way to get it fully blended will not be easy as clicking on a tool it will require manipulation of the blocks and bridge faces.

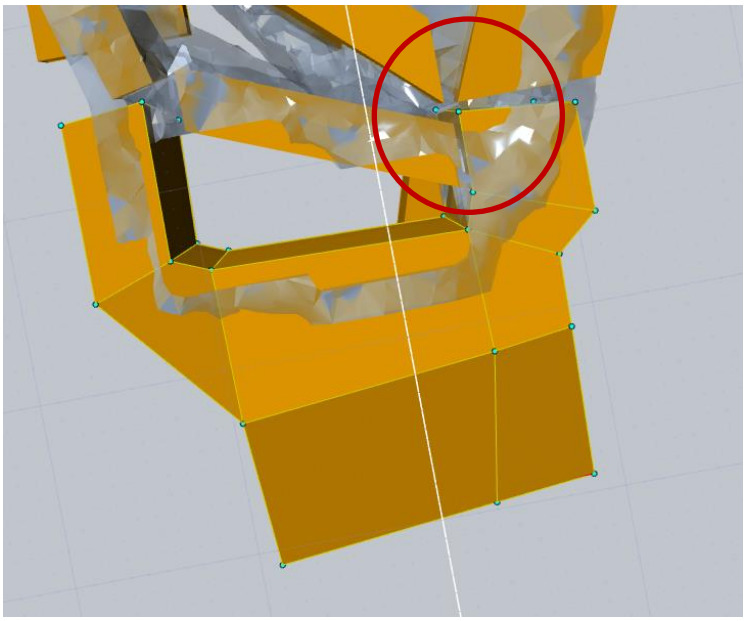


Figure 4-11: Multi Blend-Four blocks coming together

4.3.5 PolyNURBS Soft Blend

Deleting is not recommended but sometimes it is helpful, in the example of Figure 4-11 the bottom blocks are bridged and take the form in Figure 4-12. The bridging tool will just create four connecting walls in between them. These walls are not what is desired, the idea is to have the block connect directly and to do this all four edges must be deleted and take the form in Figure 4-13. When deleting in EVOLVE, the software tries its best to keep the polygonal in one piece and deleting a point, edge or face can create discontinuity problems if the wrong features are deleted. The process of bridging and deleting is called a soft blend.

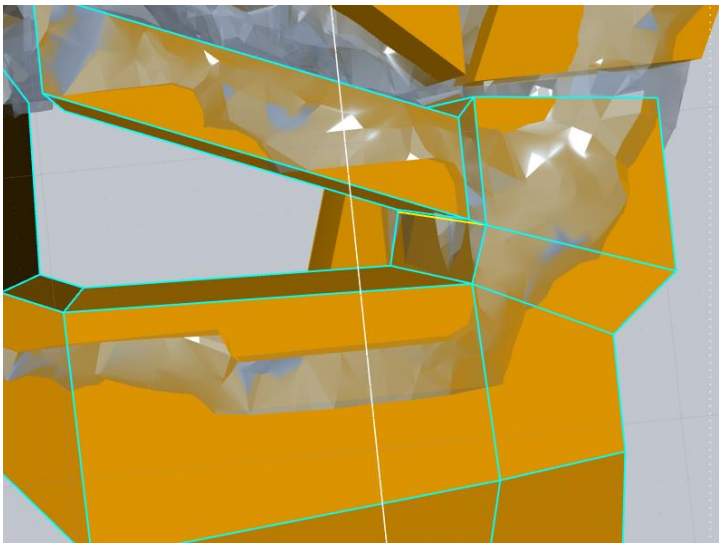


Figure 4-12: Bridge of a multi blend

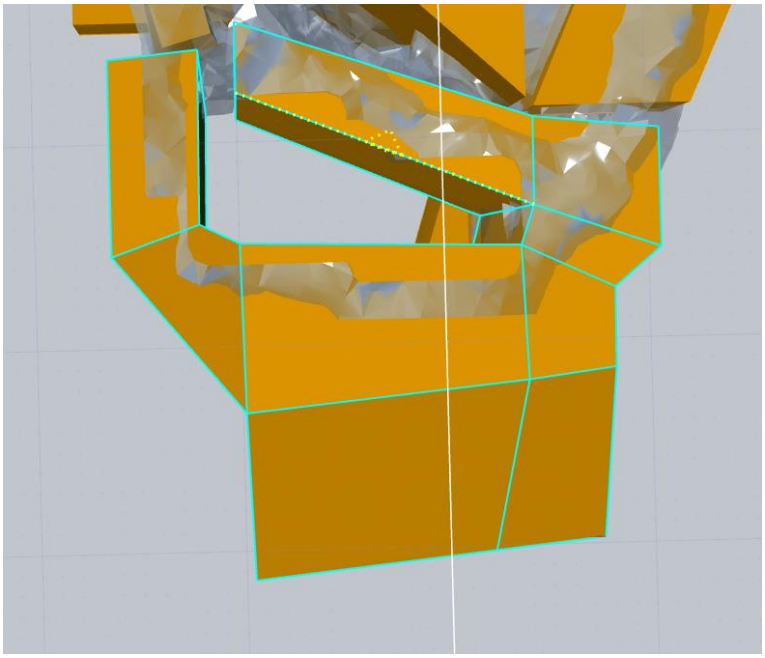


Figure 4-13: Deleting the edges of the bridge (soft blend)

In the next step is to bridge another block to the main polygonal, it is important to have a good idea on how the next block will connect for main polygonal. The top left block and the bottom block was also section cut as seen in red outline in Figure 4-14 to allow the four edges to connect. In Figure 4-14 shows the top left block connecting to the main polygonal by the use of soft blend. Another point to make is if the edges had not been deleted, the blend would not have connected at the desired point.

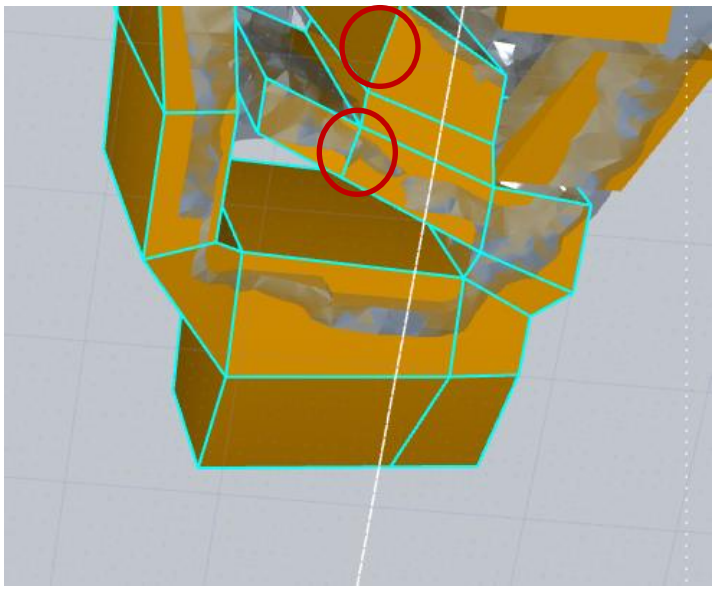


Figure 4-14: Soft blending another section

4.3.6 PolyNURBS Hard Blend

In Figure 4-14 the block on the top right needs to blend to main polygonal but will do so differently than previous blocks. The difference is the need to blend not just on one face but needs to blend two faces instead. This type of bridging is called hard blending. In EVOLVE this style of blend is not possible since bridging cannot be done with adjacent

faces. It is also important to keep the bridge connections with four edges as to avoid self-intersection [35]. Once again manipulation of the blocks so it can soft blended one at a time will be required. After rearranging the blocks and making extra cuts in the main polygonal and top right block Figure 4-15 shows all the different members come together. Hard blending will start with by having a space or gap created between the adjacent faces, this is done by section cutting. Then the opposite faces are rearranged to have easy connection to create the soft blend faces. Another example is shown in Figure 4-16, extrusion and manipulations of the main polygonal is done such that there is gap in between the two faces that will be soft blended. In Figure 4-17 the extrusion is cut to have matching faces to be hard blended, it is also important to blend the lower faces fist. In Figure 4-18 it shows the hard blend outlined in red and Figure 4-19 shows another hard blend that completes the multi blend. The multi blend is a combination of bridging, soft blends and hard blends.

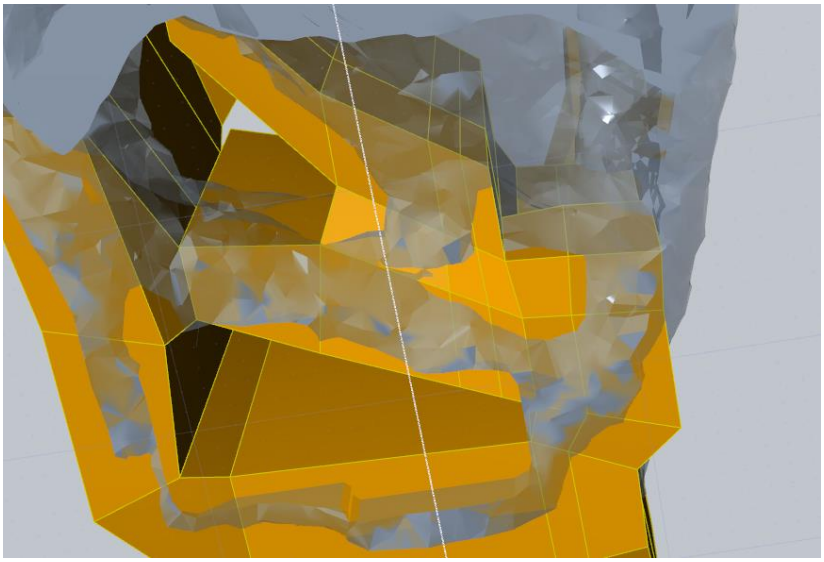


Figure 4-15: Multi blend Completed

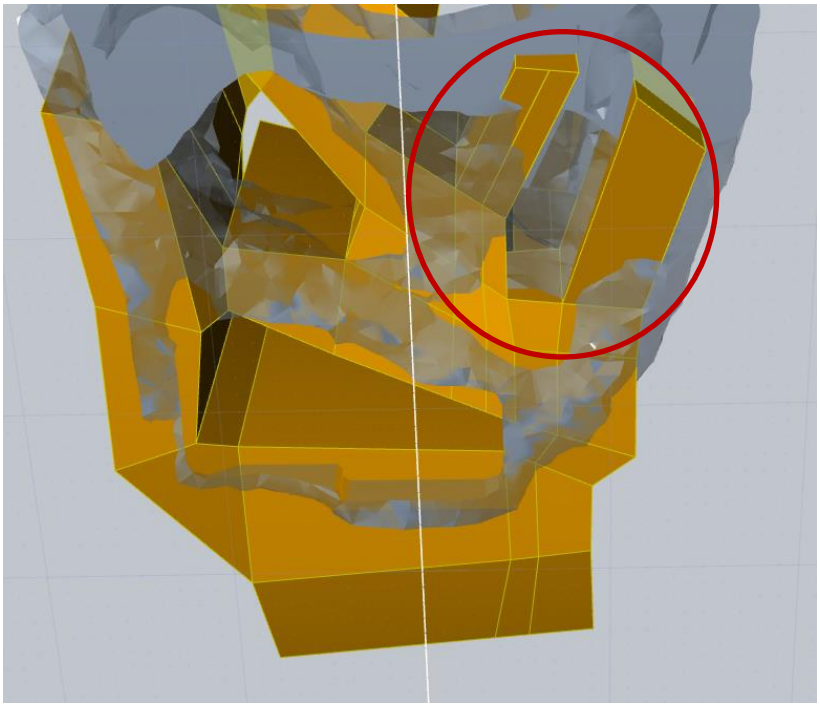


Figure 4-16: Extrusion of faces

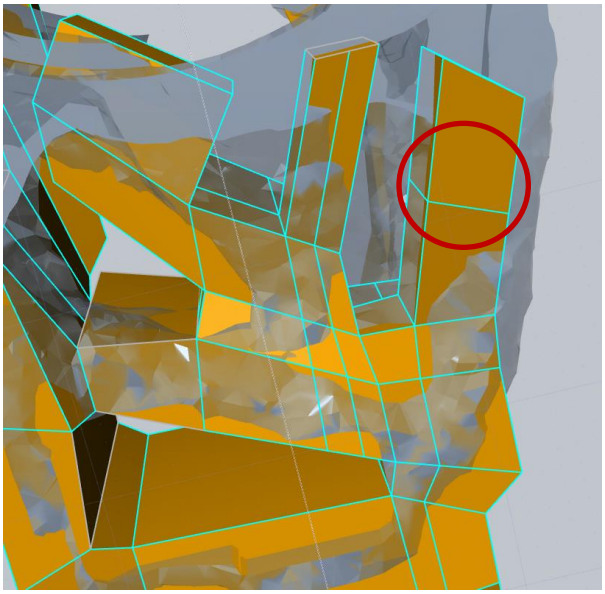


Figure 4-17: Cutting block

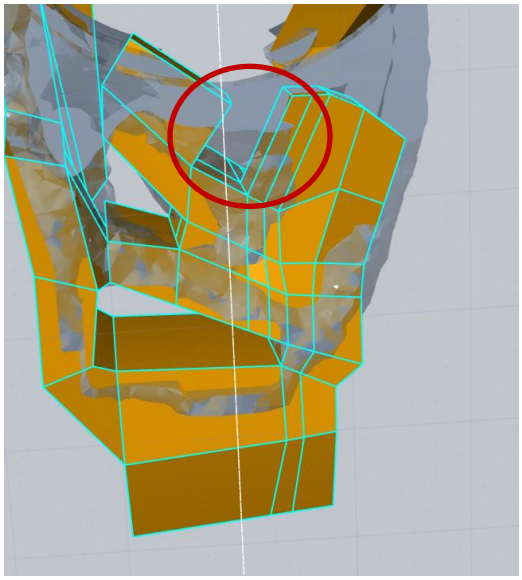


Figure 4-18: Hard blending

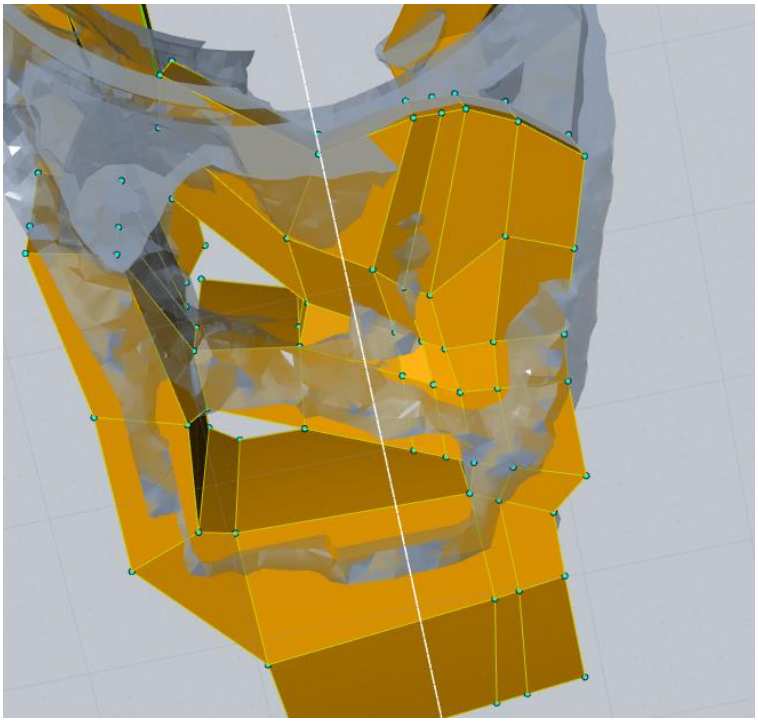


Figure 4-19: Hard blending to complete the multi blend

4.3.7 PolyNURBS Completion

With the techniques and tools explained previously it is possible to complete the rest of the topology tracing. Knowing when to use the different styles is key and it would be good idea to plan how the different blocks will come together to save time. It is recommended to have the entire the main polygonal and the rest of the block be one polyNURBS. The one polyNURBS will make it easy have all the features scaled and properly blended. Another reason for one polyNURBS is the more blends, blocks and sections the more memory it uses which slows the computer down. Lastly before moving on to the next step it is a good idea to make any final adjustment to the points, edges or faces of the one polyNURBS. When making final adjustment try and have the one polyNURBS be oversized were non-design areas are present. After this step it becomes very difficult to adjust these features as it can crash the software since rendering is happening constantly.

4.4 Finishing Modeling

Once all the work is completed with the polyNURBS then the non-design is imported as seen in Figure 4-20. If non-design being imported was the same as in the topology optimization then the non-design will overlap just fine as shown in Figure 4-21. Another thing to point out is in Figure 4-21 the polyNURBS have been greatly thicken in the center of the upright, this is a ok for it will later the trimmed off.

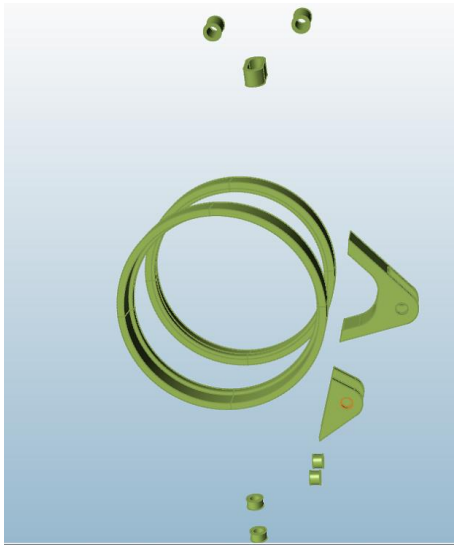


Figure 4-20: Non-design Imported

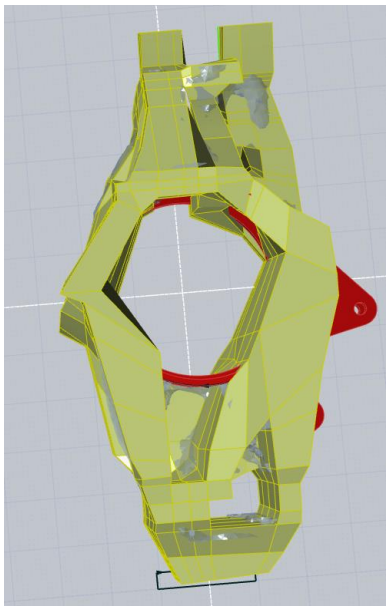


Figure 4-21: Non-design overlap

4.4.1 Trim

With non-design imported it is important to get this parts trimmed into the polyNURBS that way the polyNURBS keeps the same no-design area as seen in Figure 4-22. This non-design areas will have defined dimension and location on the model such as holes. With this information surface curves can be created to capture and use with different tools. These curves are called edge extraction and will create a curve of the outer surface of each non-design area. Some sections of the upright need to be taking out but are not necessary non-design, this spaces can be for nuts or connection to other part. The last area that needs to be trim is the outline of the part to give it flat section and give a clean look. This areas can be trimmed with a plain NURBS curve or combination of lines as shown in Figure 4-23 and Figure 4-24. Each trim line should be trimmed accordingly to match the topology results, after all the trim have been completed the polygonal can be Nurbify, the model start to get a nice finished look as seen in Figure 4-25.

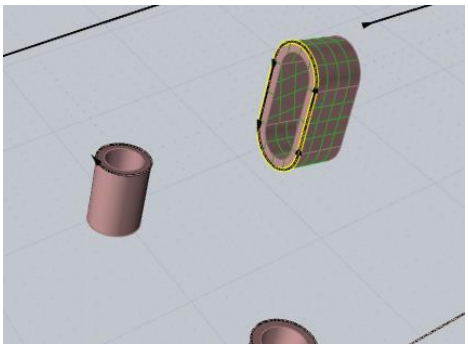


Figure 4-22: Edge extraction for trimming

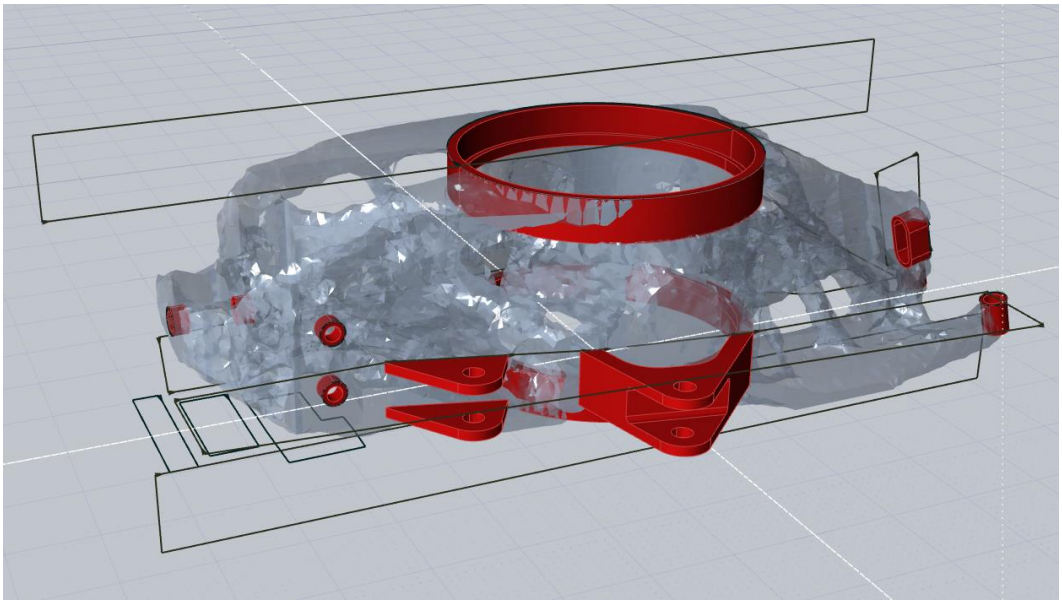


Figure 4-23: Trim lines with non-design

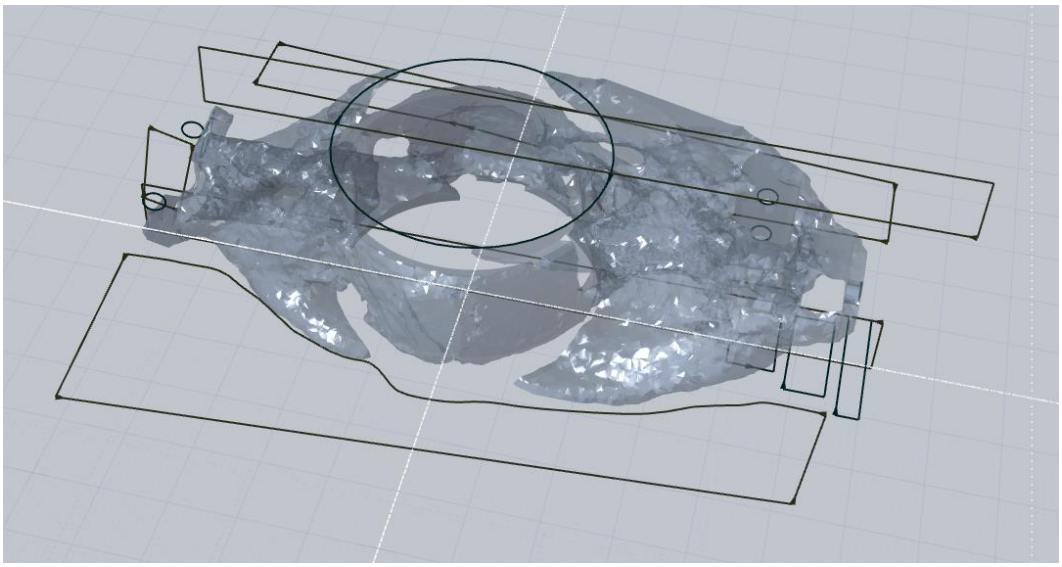


Figure 4-24: Trim lines also for the outside edges

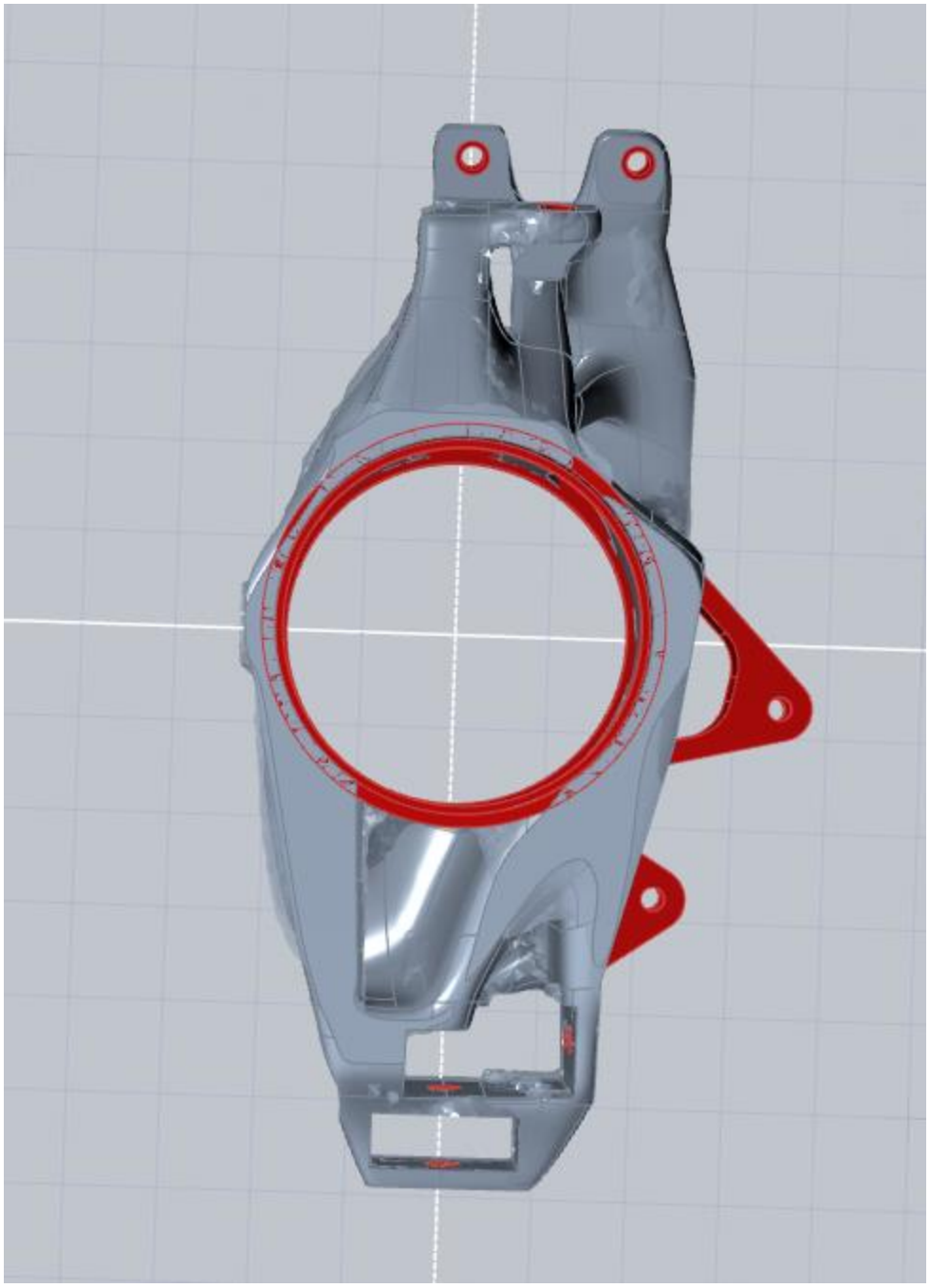


Figure 4-25: Trimmed model

4.4.2 Rounds

When the model is Nurbify it gives the option to create a solid model of the polyNURBS with all the complex surfaces and trims. The next thing to do it to perform Boolean union between all the polyNURBS and non-design. This step will make all the parts into one solid and with the trims lines being bases on the non-design curves there will be no issue Boolean as seen in Figure 4-26.

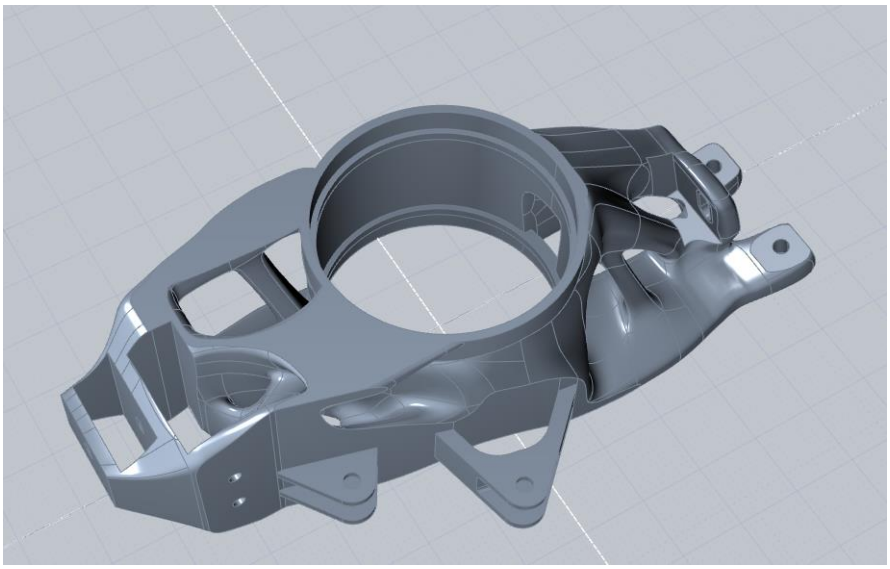


Figure 4-26: Boolean Union

The last modification that needs to be done on is to remove any sharp edges in the model and also improve the look of the model by having smooth transition between the polyNURBS to the non-design. Rounding is the term for removing the sharp edges and the only thing to look out for it that the trims form before don't create any discontinuities. Another thing to look out for is the size of the rounds need to be reasonable since the topology results can have very differently along an edge. After all

the rounds have been applied as seen in Figure 4-27 the model can be saved as a .step file and packaged to work on any other CAD or FEA software. In the case of 3D printing the model needs to be a watertight .stl file, so the polyNURBS needs to be tessellated then saved as a .stl file, as shown in Figure 4-28.

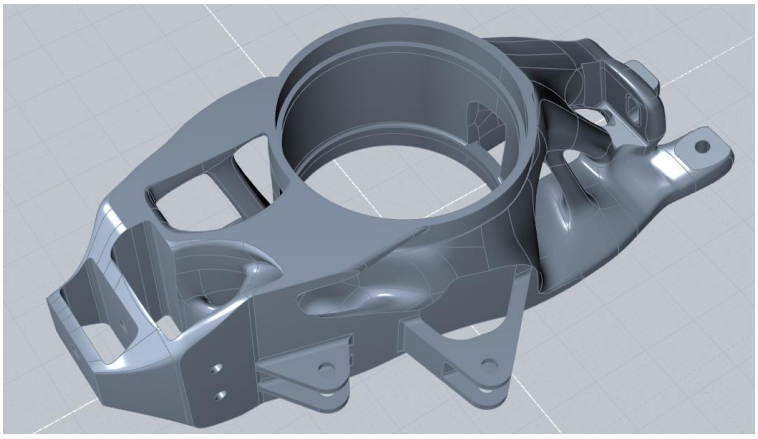


Figure 4-27: Rounding and completed solid model

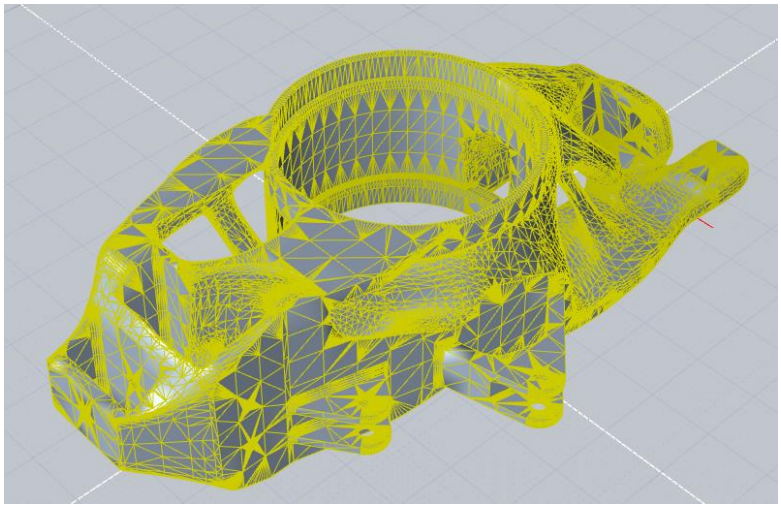


Figure 4-28: Tessellated model

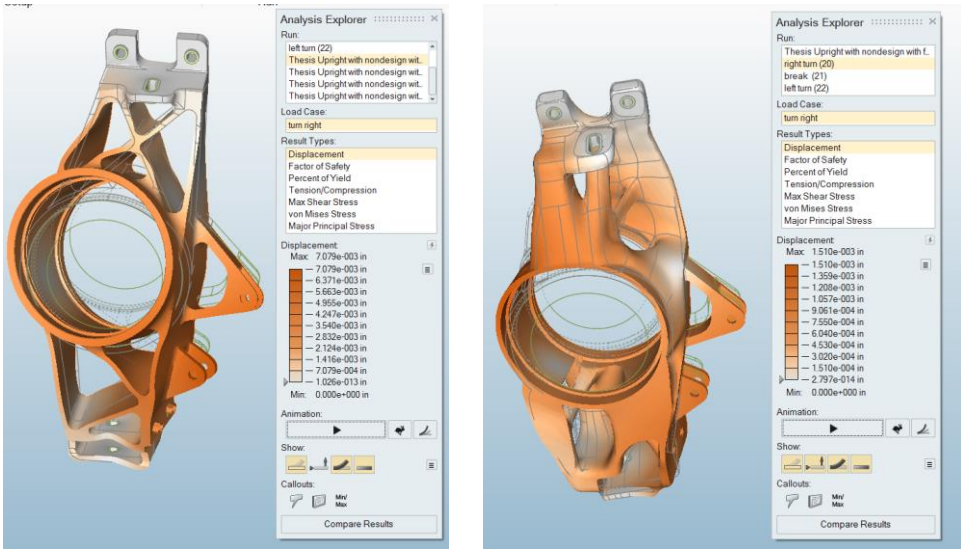
Chapter 5

Results

The solid model that was created in EVOLVE can be exported to its brother software INSPIRE. The big benefit from this is all the loads, supports and constraint are still saved in INSPIRE. If the non-design area stay the same then all the condition can be applied to does non-design area on the imported polyNURBS model from EVOLVE. To do this just delete the old design space and do subtraction Boolean from the imported polyNURBS. The tool for subtraction will be the non-design area and since this area is the one with all the information.

5.1 Analysis

With all the load case applied to the polyNURBS model it can go through FEA and show the displacements and Von Misses stress. To have an understanding on how each load case impacts the upright, analysis is run on all load cases. In the load cases of turning right and turning left there is also the load case for breaking that is added to both load cases. This inclusion of breaking load case is to show the upright doing a hard turn and breaking at the same time in both directions. The loads magnitude and direction are explained in chapter three section one of this paper. The goal of the topology optimization is to minimize compliance and to demonstrate the improvement in displacement the original and new upright are compared. The results can be seen in Figure 5-1 with a) original upright & b) polyNURBS.



(a) (b)

Figure 5-1: (a) Original upright and (b) PolyNURBS upright

5.2 Comparisons

Table 5-1 shows the results of all the runs comparing three different upright. The thesis upright -- is the one made for this paper and used PolyNURBS to create. The senior design upright -- using only CAD tools with a topology design input and no surface modeling. The original upright -- by UTA FSAE team and was made using only CAD tools, this model has a manufacturing constraint were as the other two did not. The results show the thesis upright is 0.6 lbs. more than the original. When it came to displacement it can be seen that thesis upright did reduce the displacement which was goal of the topology optimization. The results also show that turning left sees the most stress on right side upright. It is noted that the safety factor of the thesis upright is highest. Another thing to point out is the time it took to create this models, thesis using

the polyNURBS took about 160 hours were as the senior design upright only using CAD tool took 220 hours. It is also important to see the complexity of the models and how the CAD tools are just not enough to model complex geometry as show in Figure 5-2.

Table 5-1: Large Table in Landscape Orientation

	weight(lbs)	Right Turn	Left Turn	Brake	Safety Factor	Time			Total
					35 ksi	Design	Modeling	Analysis	
Thesis Upright	2.47	2.61 Ksi	13.3 Ksi	8.65 Ksi	2.63	40 hours	80 hours	40 hours	160 hours
		5.20 e-.004 in	1.51 e-.003 in	2.52 e-.003 in					
Senior Design Upright	1.73	4.61 Ksi	20.0 Ksi	11.3 Ksi	1.74	40 hours	160 hours	40 hours	220 hours
		1.28 e-.003 in	7.18 e-.003 in	3.28 e-.003 in					
Original Upright	1.87	4.05 Ksi	20.1 Ksi	10.1 Ksi	1.74	140 hours	20 hours	80 hours	240 hours
		1.25 e-.003 in	7.07 e-.003 in	3.12 e-.003 in					

Figure 5-2: Tessellated model

Chapter 6

Conclusion & Future Work

This paper takes the case study of a design optimization of a race car steering knuckle for additive manufacturing. Background information into the field of topology optimization, NURBS and additive manufacturing is reviewed to better understand the case study. The topology optimization of the upright model with the objective to minimize compliance under three different load cases and volume fraction constraint is performed. The three different load cases are turning left, turning right and hard braking. The optimization results show the load paths and high density spaces were material is needed. Figure 3-17 is the results of the final optimization run showing the complexity of topology results and the noise it can generate.

The topology results were taking to a CAID software to trace the results to create a solid model. The use of surface tool were used to create a complex model. The use of PolyNURBS was very important to generate the final model. This paper also introduced different techniques to generate some of the complex blends that are generated in topology optimization. The models is prepared for AM and also for use in other CAD software.

Analysis is ran on three different model show their displacement and von misses stress for each of the three load cases. Table 5-1 has all the results that were performed and it can be concluded the thesis upright minimized its compliance compared to the original upright. The thesis upright also took less time to complete compared to the senior design upright. It is important to see the quality that is seen in the CAD model vs. the PolyNURBS. The CAD models do not have enough capability to create a 1 for 1 model of

the topology results. The drawback of the thesis upright is the increase in weight which hurts the performance of the car.

This paper only looked at topology optimization to generate a final model. Topology optimization is a good start to structure optimization for it defines the structural features. The next step is to do shape and sizing optimization. There is more work in reducing the weight by modified the PolyNURBS. Also a look into hollow members or even lattice structures to reduce the weight with small impact to the overall stiffness of the part. Research into EVOLVE and INSPIRE future update may introduce new tools and features to help with creating complex geometries. After all forms of optimization and final model is completed structure analysis should be used and compared. If the final model show improved performance over the original upright then it should get manufactured and physically tested.

Appendix A

Figures and Analysis Results

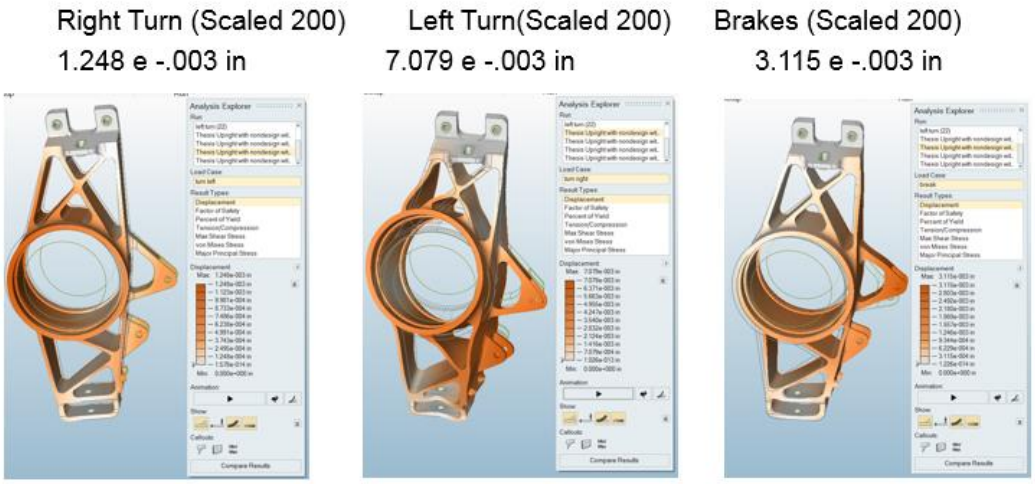


Figure A-1: Analysis Displacement Results of Original Upright

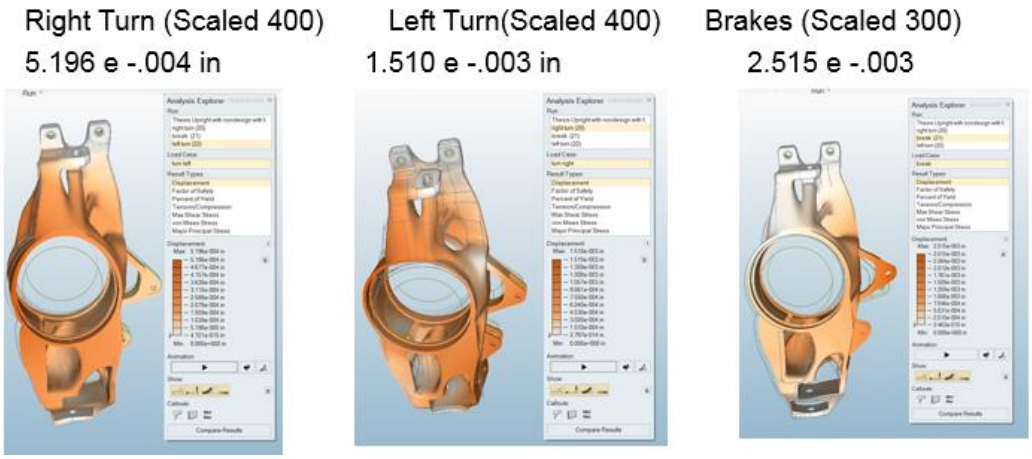


Figure A-2: Analysis Displacement Results of Thesis Upright

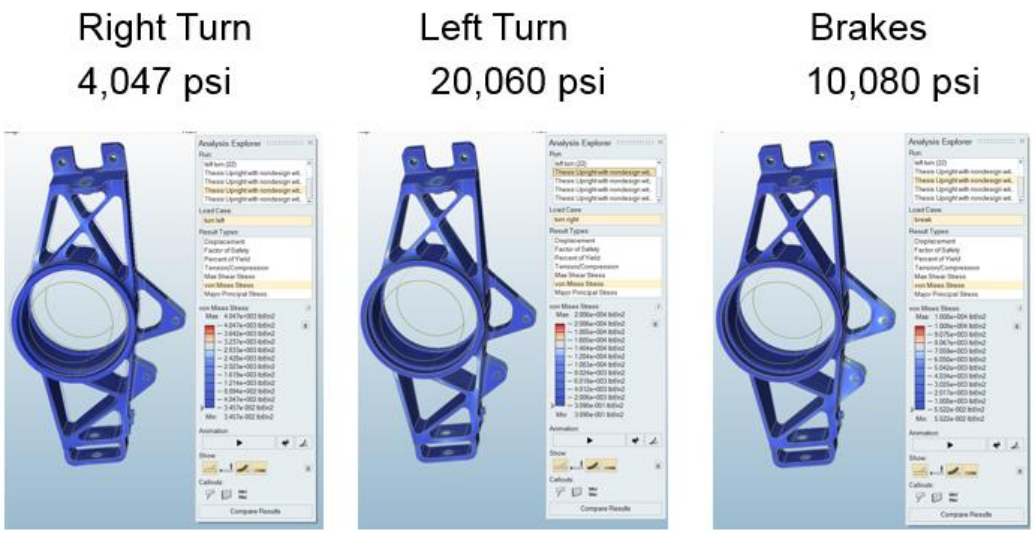


Figure A-3: Analysis Stress Results of Original Upright

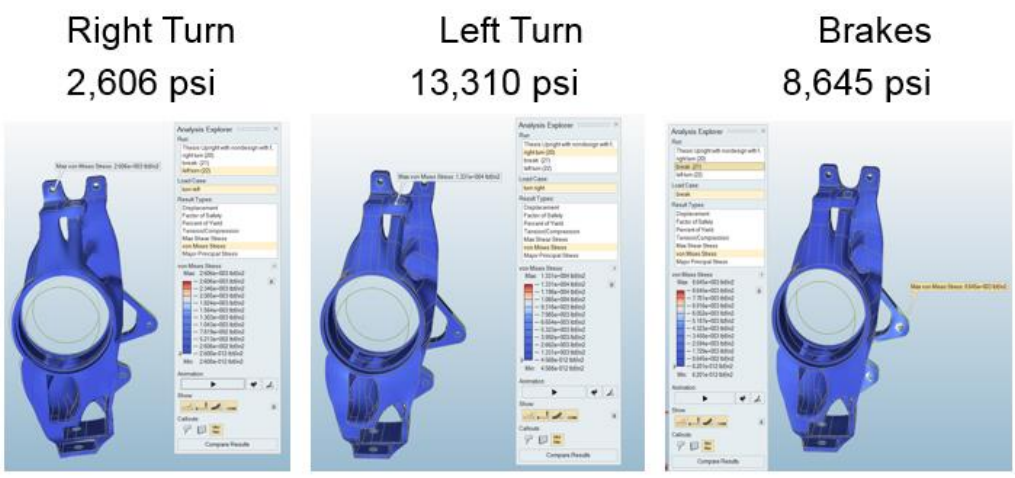


Figure A-4: Analysis Stress Results of Thesis Upright

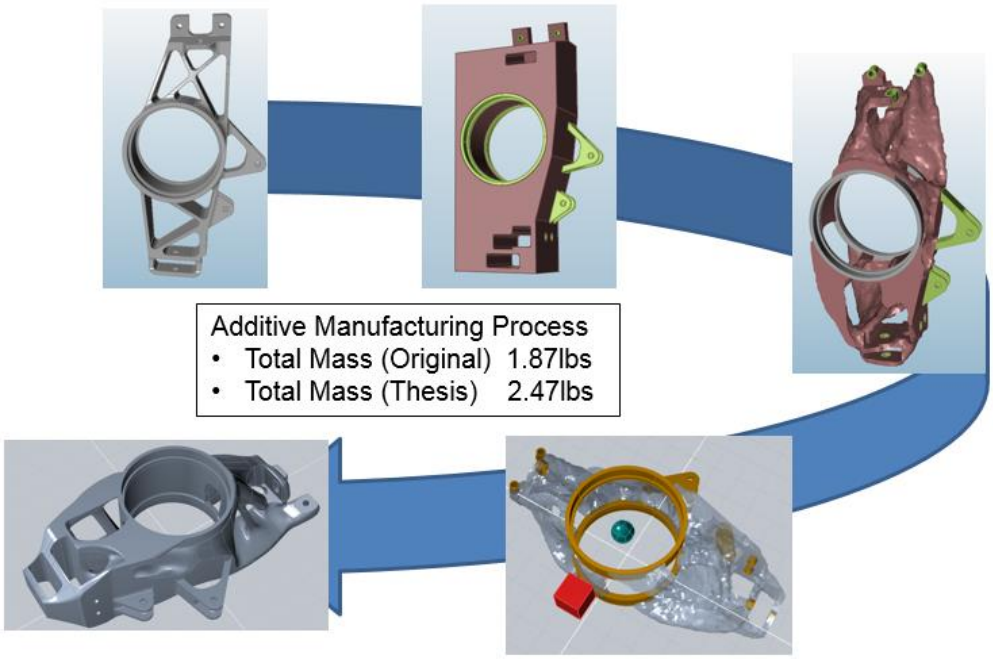


Figure A-5: Design Flow of Thesis Upright

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Biographical Information

Yobani Martinez was born in Dallas, Texas , USA in 1991. He received his high school diploma from South Grand Prairie High school in 2009. In 2011 he received his Associates degree in Science with an emphasis in Math from Mountain View Community College. He received his B.S. degree from University of Texas at Arlington, Arlington, Texas, USA in 2014 and his M.S. in 2017 from the same school, all in Mechanical Engineering. In 2016, he joined Elbit Systems of America, as a mechanical engineer doing designs then worked on structure analysis for both static and dynamic analysis. His current research is on metal additive manufacturing and structure analysis. He is a member of the UTA Rover Society Robotics.