5-1-2018

ALTAIR HYPERWORKS OPTIMIZATION DESIGN GUIDE: TOPOLOGY, FREE-SIZING, FREESHAPE, AND TOPOGRAPHY OPTIMIZATION

Alexandra Kessler

ALTAIR HYPERWORKS OPTIMIZATION DESIGN GUIDE: TOPOLOGY, FREE-SIZING, FREE-SHAPE, AND TOPOGRAPHY OPTIMIZATION

by

ALEXANDRA KESSLER

Presented to the Faculty of the Honors College of

The University of Texas at Arlington in Partial Fulfillment

of the Requirements

for the Degree of

HONORS BACHELOR OF SCIENCE IN MECHANICAL ENGINEERING

THE UNIVERSITY OF TEXAS AT ARLINGTON

May 2018

ACKNOWLEDGMENTS

A special thanks goes to Dr. Fernandez, who suggested the creation of this design guide and to Dr. Taylor, for reviewing and taking the time to explain these new concepts mentioned here. This Guide would not have been possible if not for their guidance.

I could not have done this project without the help and moral support of my fellow team members. Ryan Buckingham, Isaac Davis, Matt Leidlein, Nicholas Lira, and Stephanie Luong have my deepest thanks, and I look forward to seeing all of you on the other side of commencement.

A special thanks goes to the University of Texas at Arlington and the UTA Honor's College, particularly Bobbie Brown for assisting with formatting this guide. I appreciated your continued support in making this Design Guide possible.

In addition, a thank-you goes to the Altair HyperWorks, which provided their software free of charge to UTA so that students may explore and understand the fundamentals material optimization.

May 4, 2018

ABSTRACT

ALTAIR HYPERWORKS OPTIMIZATION DESIGN

GUIDE: TOPOLOGY, FREE-SIZING, FREE-

SHAPE, AND TOPOGRAPHY

OPTIMIZATION

Alex E. Kessler, B.S. Mechanical Engineering

The University of Texas at Arlington, 2018

Faculty Mentor: Robert Taylor

This guide is a companion paper to *2018's Optimization of the Internal Topology and Sizing for Thin-Walled Aircraft Structures – A Design and Production Guideline for Fused Deposition Modeling* and covers the methodology behind optimization in HyperWorks, specifically: topology, free-sizing, free-shape, and topography optimization. The guide includes a comprehensive summary of anything that someone attempting an optimization within HyperWorks for the first time would want to know, such as guidance, meaning, and uses for different tools mentioned throughout the guide. In addition, the benefits and drawback of structural optimization are covered. This guide is not meant to replace a basic understanding of navigation within HyperWorks and requires a respectable amount of computer and engineering acumen. The guide seeks to

define exactly how to initiate an optimization and why. Lastly, the guide will reveal in depth the justifications of beam member placement during topology optimization of the fuselage and a supplementary free-sizing optimization for possible future use by the next senior design team.

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1 Structural Optimization

Before the advent of computers, structural and mechanical engineers relied on a combination of brute-force mathematics and trial-and-error to accomplish more streamlined, lightweight, and effective structures. This was time consuming work and did not yield optimal results. Now in the 21st century, the ability to solve complex problems, such as structural optimization with computational algorithms using finite element analysis (FEM), has accelerated part analysis, revolutionized the way we manufacture components, and deepened our understanding of the design process.

Structural optimization is the automated process by which a less effective part can be made more streamlined through the minimizing or maximizing of a response from the model such as minimizing the displacement of a node or the volume of the model. For an effective optimization, the user must have a clear understanding of how the part is constrained and what forces are acting on it. Making a clear free body diagram (FBD) with all parameters, including forces, the reaction points, material information, constraints, and design spaces is essential.

HyperWorks is a fully operational computer aided engineering software (CAE). The product includes a modeler and mesher (HyperMesh) as well as an optimization solver (OptiStruct), which uses the gradient method for optimization. HyperMesh is used mainly in the aerospace, architectural, and automotive industry to remove unneeded weight and increase efficiency, but has been applied to other industries, such as those with 3D printing applications, to great success. There are 6 main optimization types available in HyperWorks, including Topology, free-sizing, shape, size, free shape, topography, and composite shuffle optimization. The optimizations that will be covered in this guide include topology, free-sizing, shape, and topography.

1.1.1 Advantages of Structural Optimization

Structural optimization carries with it many advantages, which can lower the cost and maximize the effectiveness of a part. Topology of a structure, for example, generally leads to less material usage, and therefor less material cost. Less material usage also leads to a lighter part. This is key in the aerospace industry, where every pound that is removed from an aircraft increases the efficiency of the vehicle. In addition, optimizing with maximizing stiffness (or minimizing compliance) achieves the lightest part with the strongest body. This was done in the 2018 Guide to minimize the effect of material removal. Optimization can solve issues such as fitting a part into a slot without reducing the part's overall strength. In addition, designers can go back in at any point in time, review and edit the optimization, and rerun the results quickly to get a new result. This is a major timesaver if in the past, the alternative was to redo the math, manufacture, and test the part again.

1.1.2 Disadvantages of Structural Optimization

While optimization has numerous applications and endless benefits, it does not benefit in *all* aspects. First, it must be certain that there is a need to optimize. Depending on the part's application and the business, optimization may not be the right solution for reasons that will become apparent. Optimizing too early in the design phase or when it is not beneficial would be costly or time consuming. As the prominent programmer Donald Couth stated, “Premature optimization is the root of all evil”. Though perhaps a bit too on the nose, his statement provides some insight as to the relationship between professionals and the concept of optimization early in the design process.

After optimization, the geometry of the part is almost always more complex. The machinery required to bring that geometry to life may not be within the abilities or budget of the user. For example, Figure 1.1 shows both the initial, optimized, and final geometry involved in an optimization.

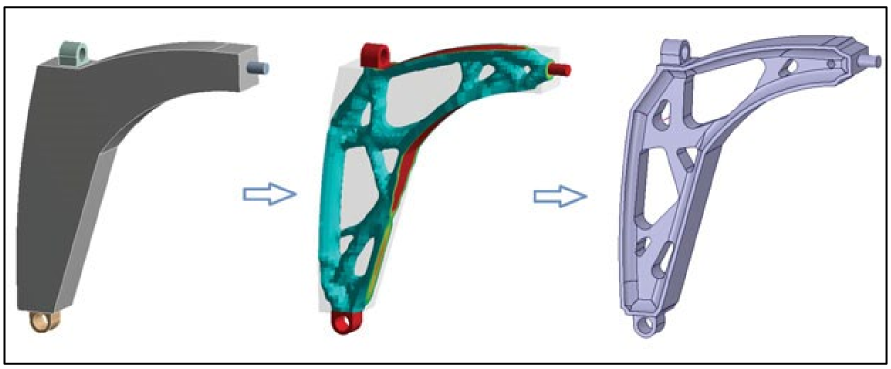


Figure 1.1: Optimized Automobile Upper Control Arm (Bhate)

What would have taken a simple cut on a thicker sheet of steel now requires at least a 5-axis mill and a part-flip between machining. Precious time is lost, and the appropriate machine must be acquired to even begin the process.

The cost of designing the optimize part as well as the time and assets required to print must be considered in any business situation. A computer, to run the optimization, is needed. The system requirements on this computer will only increase as the geometry, solution, or file components become more complex. Once the optimization is done, the objective may be to edit the part further. Since the geometry is more complex, more time would need to be taken to find problem areas and the solution to these problem areas in any type of solid modeling interface such as CATIA or SolidWorks may be more involved. In this case, time most certainly equals money.

Once a part area is optimized, the optimized solution will most likely leave little room for manufacturing defects. For example, in a topology optimization, a solid part might have a safety factor 3 times more effective than an optimized solution since the optimized solution is attempting to find the most effective material layout will the least amount of material.

Lastly, and possibly most obviously, is the necessity of apply design constraints and parameters correctly and having an impeccable understanding of the program itself. It is very often noted that when optimization programs, or any FEA programs, are used incorrectly, it can have disastrous results, leading to failing or sagging parts, or causing the designers themselves to start back at nearly the beginning of the design process.

1.2 HyperWorks

HyperWorks 2017 is a finite element analysis (FEM) software capable of optimizing a structure based on a specified objective. To solve, the software optimization is limited by a constraint and its objective. Within those boundaries, the software will work iteratively until it meets the constraints and has minimized or maximized its objective consideration. HyperWorks is a unitless solver, so care must be taken to make sure that all information is inputted with the same units in mind. Before the optimization of the model takes place, the model must contain the material, properties, loads, load steps, objective, constraints, and responses necessary to fully define the problem to be optimized.

The optimization solver, OptiStruct, solves the optimization by using finite elements such as squares or triangular surfaces to approximate a model and more easily solve for the stretch and pull of these simpler shapes. In particular, this method is called the Gradient Method. This method can be simplified mathematically to:

$$\varphi_0(p) \Rightarrow \min(\max) \quad (1)$$

$$\varphi_i(p) \leq 0 \quad (2)$$

$$p^l \leq p^j \leq p^u \quad (3)$$

Equation (1) represents the objective function, where it attempts to minimize or maximize a variable. Equation (2), which is related to equation (1) through variable association, keeps other variables below a certain threshold. Equation (3) defines the design space in which the optimization occurs, where p^l and p^u represent the uppermost and lowermost bounds.

1.2.1 Model Meshing (Components)

HyperWorks requires the use of a mesh for all of its functions. A mesh is a collection of elements based on a surface or 3D IGES or STEP model. The resolution of this mesh often is directly proportional to the quality of the final product. A low-resolution mesh will average the forces occurring in an element and result in an optimization solution that cannot reveal any beam members, thickness, or deformation information accurately.

1.2.2 Material Card

A material card defines properties such as Modulus of Elasticity (E), Shear Modulus of Elasticity (G), Poisson's ratio (Nu), density (Rho), thermal expansion coefficient (A), and a slew of other properties.

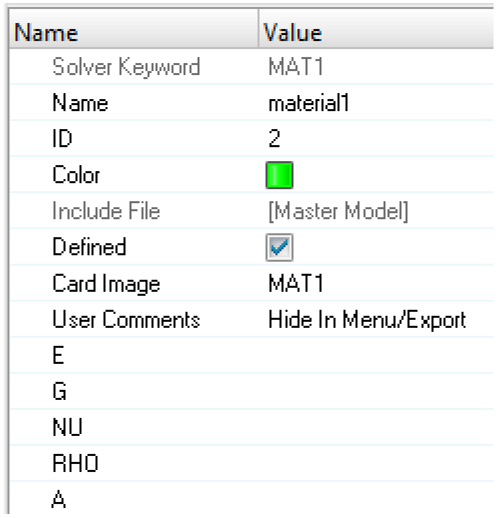


Figure 1.2: Material Card

An example of a material card is shown in Figure 1.2. Notice that these fields will not self-populate and need to be addressed. Here, it is important to review the units of the model, as incorrect unit placement in this area can have detrimental effects on the rest of the optimization.

1.2.3 Properties Card

A property card is used to define the thickness of the material for the surface mesh. Here, the thickness of the surface mesh is defined. This thickness is the thickness of the part before optimization. This would not change in topology optimization, as the thickness remains the same, but is something to keep in mind when doing topology optimization and sizing optimization because the program may either increase or decrease this parameter afterwards.

1.2.4 Load Collectors

Load collectors represent the method of organizing separate loads within HyperWorks. A load collector can carry several loads, be that constraints, moments, forces, pressure, or accelerations. For loads to be applied correctly, the user needs to know precisely where, what kind of load, and what magnitude is needed. Constraints within HyperMesh can pin down a node on the mesh through the six degrees of freedom.

1.2.5 Load Steps

HyperWorks associates a force and a reaction force (the constraints) to create static structural cases that can be optimized. A load case is important because it sets up the optimization to solve for each separate condition. A good example of this type of application is when you have a two-load cases on a building: one case would be when the building is subjected to wind from the north, and the fixed constraint of ground to building. Then, you would want one with wind from the east and the fixed constraint of ground to building, etc.

1.2.6 Responses

Response cards in HyperWorks are selected parameters by the user to reference the objective and the constraint of the optimization. For example, if the amount of volume left over after the final optimization iteration must be below 30%, the optimization algorithm will use the volume response after each iteration to determine its next course of action, such as minimizing the volume further to stay within its goal. Another example would be when using weighted compliance, which notes the compliance of each loading on object. These weighted compliances can help solve a problem for stiffness further on in the setup and initialization. The available responses are shown in Figure 1.3.

mass	inertia	static force	composite strair	frf pressure	function	temperature	psd strain	rms stress	gpforce
massfrac	compliance	static stress	frf acceleration	frf stress	beadfrac	psd accleration	psd velocity	rms strain	mode shape
volume	frequency	static strain	frf displacement	frf strain	compliance inde	psd displaceme	rms accleration	rms velocity	resforce
volumefrac	buckling	composite failur	frf erp	frf velocity	weighted comp	psd pressure	rms displaceme	external	thermal complia
cog	static displaceme	composite stres	frf force	fatigue	weighted freq	psd stress	rms pressure	spc force	boredst

Figure 1.3: Table of Responses Available in HyperWorks

Some material cards can only be applied to certain optimizations, such as volumefrac, which cannot be applied to a topography optimization because in a topography optimization, there is nothing that can be optimized volume-wise.

1.2.7 Constraints

Constraints are one of the ways in which HyperWorks verifies that it has reached a final optimization solution. For example, if a volumetric fraction constraint is applied, this fraction would be denoted by the Equation 1.1. If optimizing for the weight of the structure, the amount of volume left over after the final optimization iteration must be a fraction of that with which it began. A design choice can be made for the volume to have a maximum of 30% of its original value, thus:

$$volume\ fraction = \frac{total\ volume\ at\ current\ iteration}{initial\ design\ volume} = \frac{30}{100} \quad (1.1)$$

A volume fraction constraint ensures that the optimization is only keeping a fraction of the weight of the structure while the program optimizes for another objective. Through each manual iteration of an optimization, as the designer analyzes the results and refines the parameters, the volume fraction can be lowered if it seems that the structure can handle it. However, it is important not to lower it excessively. Lowering the volume fraction can create crisper, more defined members, but could also create more unreliable structures. For example, using 3D printed materials with any thinner members could lead to breakage along the direction of printing.

1.2.8 Objective

The objective card of the HyperWorks has the responsibility of minimizing or maximizing a particular response. In this way, HyperWorks optimizes the structure through iterations to reach a final solution. For example, weighted compliance weighs each load case individually then finds a stiffening structure based on the minimization of this compliance. Minimizing compliance is the same as maximizing stiffness of a structure, and thus the user can be sure that the program is doing its best to attempt to retain as much strength in each member as possible. The objective function has the most influence over what form the optimization will take.

CHAPTER 2

TOPOLOGY OPTIMIZATION

2.1 Topology Optimization Theory

Topology optimization seeks to find the optimal layout for a structure within a given package of limited volume using finite elements whose densities are adjusted in each iteration of the solution. Topology optimization of a structure generally leads to less material usage, and a reduction in material cost. However, manufacturing cost may increase if the final complex structure of a topology optimization is used as the final machinable geometry. Exporting only the topology optimization as a final solution causes many issues, both with beam members loading effectiveness and machineability, and is not advised. Instead, the final optimization solution should be streamlined in a geometry-editing software to ensure lower manufacturing costs.

The topology solution is defined using the parameters set forth by the user. More specifically, the topology parameter is defined within the design variable, which connects to the properties, the thickness of the part, material distribution parameters such as thickness between members, and member thickness itself. The results of the topology optimization are first and foremost dependent on the parameters set forth by the constraints, forces, and the objectives of the optimization. All of these must be defined beforehand. The objective of the optimization is of the utmost priority. An optimization that is dependent on the reduction of the volume of a part reveals drastically different results to that of a part that is dependent on the displacement of a selection of nodes or of weighted compliance. The topology optimization parameter, once created and property selected, looks as shown in Figure 2.1.

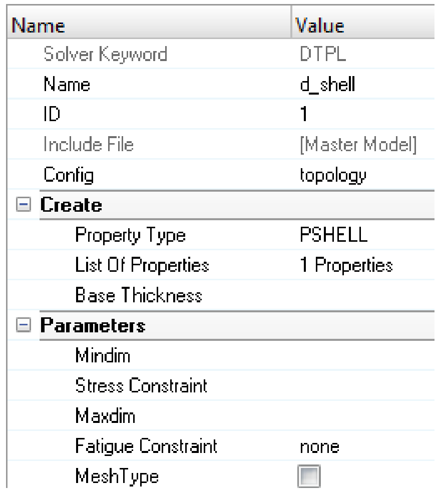


Figure 2.1: Topology Optimization Design Variable

Each section, when edited, adjusts the outcome of the optimization. The original model thickness comes from the property that is attached to the design variable. If it is not already identified in the connected property, "base thickness" is where it must be identified. Under parameters, these are what define how the member will appear once completed. "Mindim" and "Maxdim" represent the minimum and maximum thicknesses parallel to the surface of the members that result from the optimization. "Stress constraint" and "fatigue constraint" can be defined if there is a maximum stress or fatigue value on the part that should not be exceeded. Fatigue is likewise. Putting a value in for "Maxdim" suggests minimum gap size of zero that can be edited. Leaving this gap size at zero can result in beams that are too close together. Setting a minimum gap size; however, can also be detrimental if the members must connect closely, as members will be spread more thinly.

2.2 Guided Topology Optimization

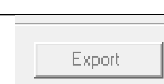
This topology optimization guide will show how to set up, mesh, assign materials, load collectors, apply a volume response, constraints, design variables, objectives and output and interpret the solution. Here, the objective is to optimize a C-Clip for volume with a requirement that it withstand two forces on its arm of 100 N in in opposing directions.

2.2.1 Surface Model Creation

Here, the model is first created in HyperWorks or a geometry creation tool of the user's choosing. The geometry is kept simple to ensure understanding. Then, the geometry is imported into HyperWorks in order to set up the optimization parameters.

1. Choose your preferred surface model creation application.
2. Create a surface model with a square that is 100x100mm.
3. At its midsection, remove from the right side a rectangle that is 20x40mm.
4. Remove a partial circle with a diameter of 20mm at the left end of the rectangular cutout.
5. The surface model can be imported into HyperWorks as an STL file.

Table 2.1: Import STL Files

Import	Solver Deck	STL	Export
			

The outline of the part should now be visible as shown in Figure 2.2.

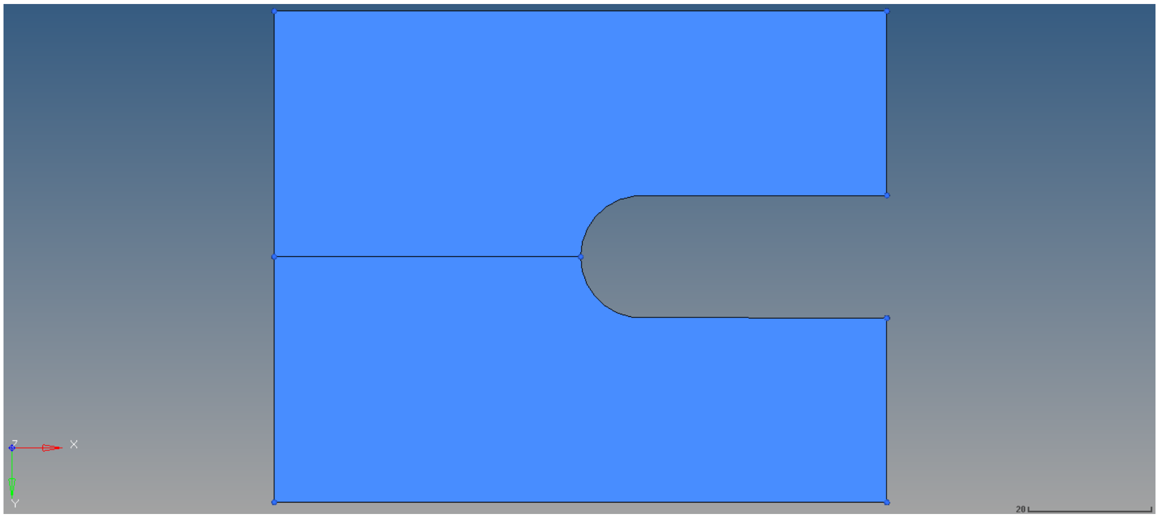


Figure 2.2: C-Clip Outline

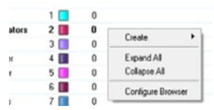
Ensure that all edges are appropriately connected by displaying the surface model and then going to geometry, then edge edit. Once this is verified, create the mesh.

2.2.2 Mesh Creation

As HyperWorks works with finite elements, the imported surface model will not work on its own, so a mesh must be created in a separate component. When formulating the mesh, it is important to select size and bias and then define the element size as well as the mesh type. This selection is dependent on the allowable element number of the program as well as the resolution needed to receive an accurate but time-efficient solution.

1. Create a new component using the card creation menu that is accessible in the model view tree area.

Table 2.2: Card Creation

Right click	Create	Select Component
	Create	Assembly, Beam Section Collector, Beamsection, Block, Component, Contact, Contact Surface, Cross Section, Curve, Feature, Field, Group, Include File, Laminate, Load Collector, Load Step, Material, Multibody, Output Block, Parameter, Plot, Ply, Property, Region, Sensor, Set, System Collector, Vector Collector, View

2. Create a mesh

Table 2.3: Mesh Creation

2D	Automesh	Displayed surfaces	Mesh
2D	automesh	surfs	mesh

3. The result should be as shown in Figure 2.3:

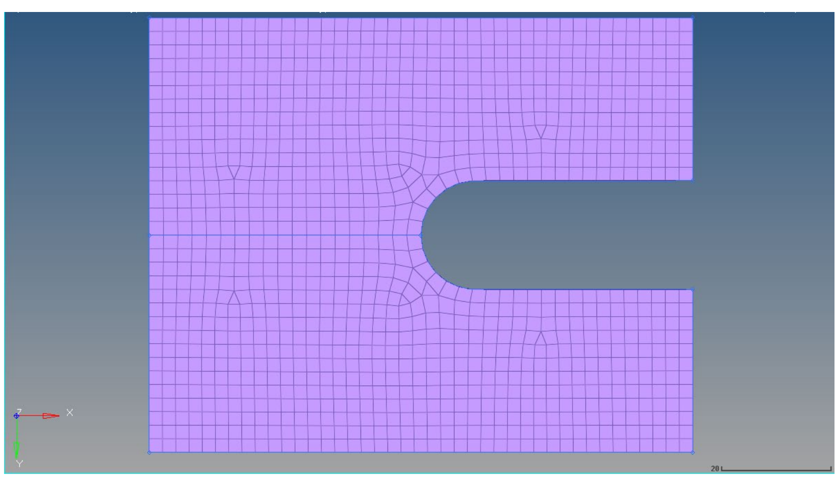


Figure 2.3: Meshed Surface

4. The original model in the components section may now be deleted.

2.2.3 Material Card

Now, the material card must be defined. In this case we can use the defaults of steel in the program. This is a similar process as was described in Table 2.2. However, instead of selecting *Component*, select *Material*.

1. Name the material “Steel”.
2. Click the blank box in E and the number will show up. Click enter to confirm the number. Do this for E (Modulus of Elasticity), Nu (Poisson’s ratio), and Rho (Density).

The Modulus of Elasticity, Poisson’s ratio, and density are the minimum number of values that must be defined. Again, it cannot be expressed how important it is to understand the units involved in the HyperWorks solver to ensure that the optimization outputs meaningful results.

2.2.4 Properties Card

Next create the properties of the surface model. In this, the thickness of the part will be defined as 1mm as well as its material referenced. This is a similar process as was described in Table 2.2. However, instead of selecting *Component*, select *Property.*

1. Name the property “prop_shell”
2. In the card, assign the material as “steel”
3. In the card, assign thickness as 1 (mm)
4. Go to the mesh component and select the new property from the dropdown menu.

2.2.5 Applying Load Collectors and Load Steps

Next, loads and constraints must be applied to the surface model. The load collectors in HyperWorks include both constraints and forces, moments, and pressures that may be applied to the model.

1. To create the full load collector entity, first create the constraints as shown in Table 2.4, then the loads using the method in the same manner. It is important to note that only one constraint card is necessary and at least one load card for proper optimization.

Table 2.4: Create Loads

analysis	Select load type	Nodes (+ dof, mag)	Create
Analysis	constraints	nodes	create

2. Three constraints must be added to the part as shown in Figure 2.4 with the degrees of freedom as labeled in the image. In essence, the part is being constrained in the y and z directions and is pinned down at its back-center point. This constrains the C-Clip to certain planes while still allowing the part to deform.

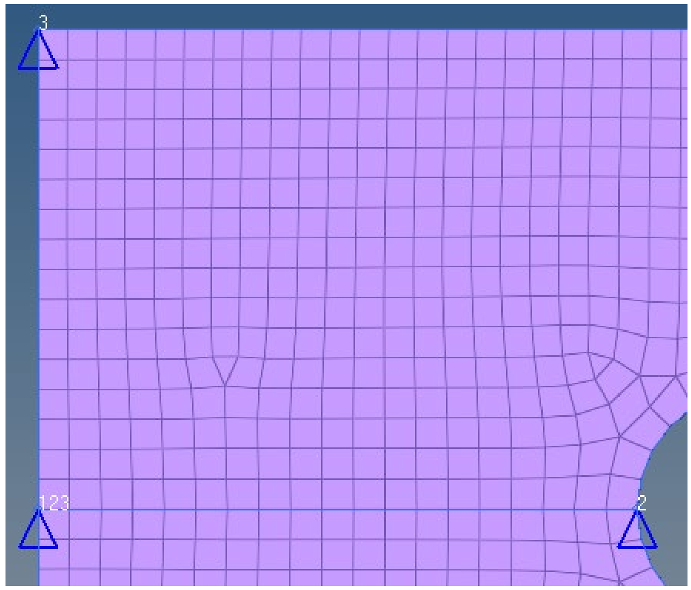


Figure 2.4: Constraints of Model

3. Make the new load collector current by the same method in Table 2.4 as a force with a magnitude of 100 N. The result should be two forces set at the edges of rectangular cutout as in Figure 2.5. Momentum and pressures can be applied in much the same way through the analysis pane.

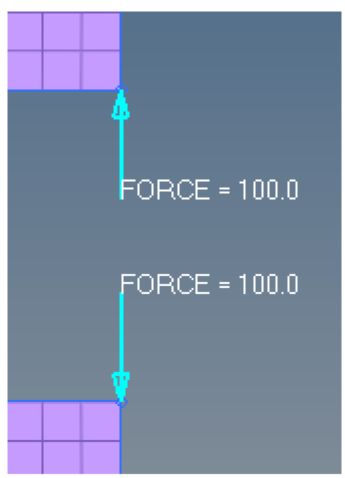


Figure 2.5: Force Applications

4. A load step for both forces can be created in a similar way to Table 2.2, but selecting *Property*, instead of *Component*. Select *Load Step* from the list. In the

card, constraints are selected as being SPC, or "Single Point Constraints", as shown in Figure 2.3-11. Both the constraint and at least one load must be defined as in Figure 2.6.

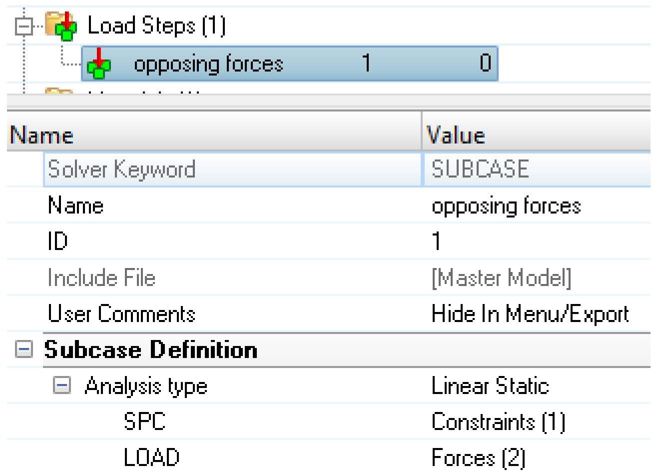


Figure 2.6: Load Step for Model

2.2.6 Volume Response

Next, a volume response in the form of a volume fraction is selected. By creating a volume response card, this can be later referenced by the program to optimize or constrain the solution.

1. To create a volume fraction response, go to analysis, optimization, responses, and input a name, response type (volfram), and enter the load step already set up. For volfrac, the total volume was used (do not select any constraint(s) for this step).

Table 2.5: Create Responses

Analysis	Optim.	Responses	Name	Response type (+total/ loads)	Create
Analysis	optimization	responses dconstraints	response =	response type	create

2. Next, create a displacement response for each of the forces (both upper and lower) on the C-Clip using the same method, but choosing static displacement, selecting the nodes from the upper force, then the lower in a separate response, and choosing dof2. Dof 2 was selecting in order to see its deflection in the in-plane direction.

2.2.7 Constraints

To create a constraint that keeps the arms of the C-Clip from deflecting, two parameters must be set as the constraints for both the upper arm and the lower arm.

1. Select topology optimization within the optimization menu and then “dconstraints”. Name the constraint and reference the static displacement response for the upper arm that has already been created. Set a value for the

displacement between two values or just select an upper or lower value. In this case, the upper constraint was chosen to be 0.07.

Table 2.6: Create Constraints

Analysis	Optim.	Constraints	Name	Range	Create
Analysis	optimization	dconstraints	constraint =	lower bound = -1.000e+20 upper bound = 0.070	create

2. Create the second constraint by selecting the lower arm's response and choosing a lower bound of -0.07.
3. Click create.

2.2.8 Design Variable

To create the topology parameter, a design variable must be created, which will determine the type of optimization analysis done.

1. Go to analysis, and then the optimization menu. In the topology menu, select a name for the new design variable card as well as PSHELL, which defines the type of model for which the software is optimizing.
2. In addition, thickness should be set to 0.0 in order to reduce unimportant areas to a density of zero. A base thickness of zero indicates that this area will turn into a void.
3. Click create.

Table 2.7: Create Topology Design Variable

Analysis	Optim.	Topology	Create (+name)	Select Properties (+PSHELL, b. thick: 0.0)	Create
Analysis	optimization	topology	create	props	create

2.2.9 Define the Objective

To ensure an optimization solution is reached, a objective card should be included that states for what the program is trying to optimize.

1. The objective can be created using the optimization menu. In order to create the objective, go to analysis, then the optimization menu and select objective. Here, the response that is to be optimized for is selected. In this case, the objective is to minimize the amount of material, or the volume, of the C-Clip.

Table 2.8: Create Objective

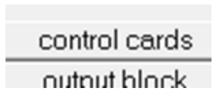
Analysis	Optimization	Objective	Select response: weighted comp. (+min)	Create
Analysis	optimization	obj reference objective	response =	create

2.2.10 Output the Solution

The optimization is nearly ready to execute. It is important that the information be outputted to the Solver View so that the user knows the progress of the optimization.

1. To ensure that the optimization outputs to the viewer, ensure that “screen” is activated.

Table 2.9: Output to Solver Viewer

Analysis	Control Cards	Next	Next	Screen	Return
					

2. The card image should now be green as shown if Figure 2.7.

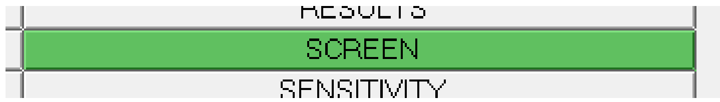


Figure 2.7: Screen Output Activated

2.2.11 Check Model and Run Analysis

The final Model tree should look similar to Figure 2.8. In the model, the component includes the surface and mesh, the design variable, both load collectors as constraints and forces, one load step, the material steel, the objective of optimizing for volume, both optimization constraints for both arms, three optimization responses for the constraints and optimization, and the properties of the surface have been defined.

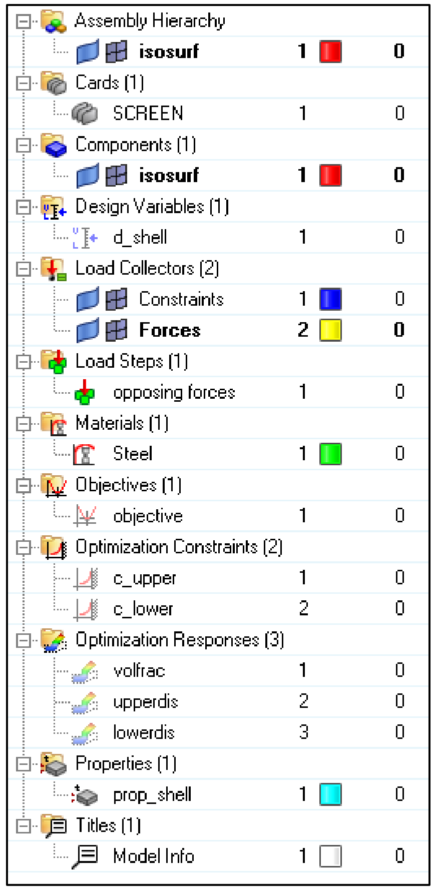


Figure 2.8: Model Tree

Then, the optimization can be run through HyperWork's FEM analysis solver software, OptisStruct.

1. This optimization can be run by selecting optimization in the OptiStruct menu. It is important to save the file as a FEM file to locate it later, as well as selecting optimization in the run options, setting the output to "all", and setting the memory allocation to default. A popup displaying the process of the optimization will the run in Solver View.

Table 2.10: Run the Optimization

Analysis	OptiStruct	Save as.. (+.fem, optimization, all, memory default)	OptiStruct
Analysis	OptiStruct	save as...	return

2. Once the Optimization completes effectively, the solver view displays as shown in Figure 2.9 as a feasible design solution with a process time of roughly 5 minutes and 28 iterations.

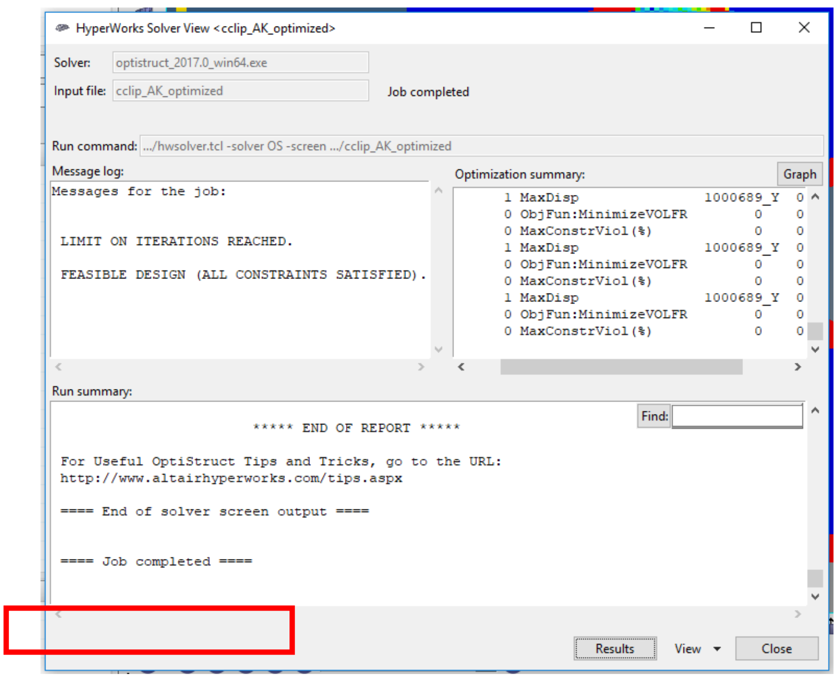


Figure 2.9: Optimization Solution Completed

3. The results can be viewed through HyperView right clicking and plotting the results. Figure 2.10 shows the results of the C-Clip optimization.

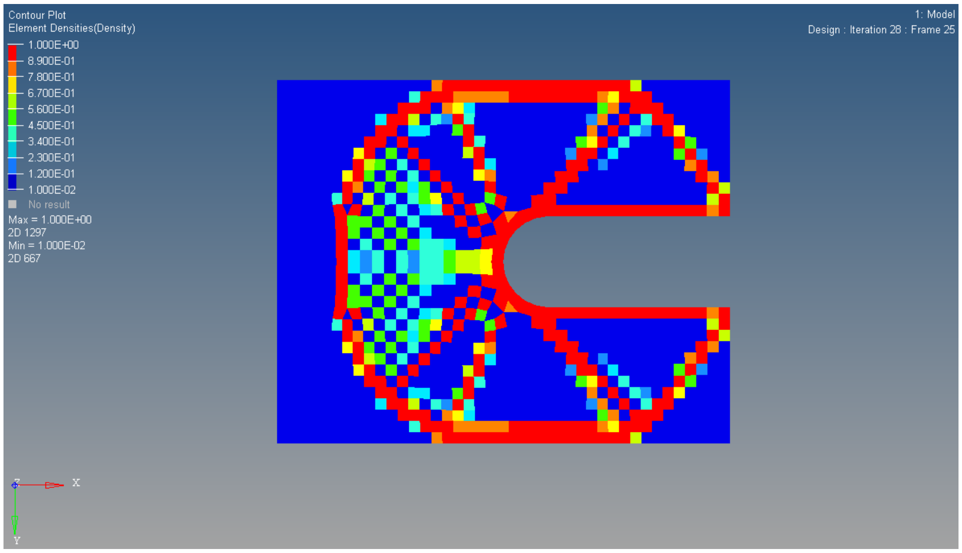


Figure 2.10: Optimized Solution Heat Map

1. Notice the results are pixelated due to the low number of elements being used. The results can be refined by cutting the mesh finite elements in half in the HM file through HyperMesh. The refined solution is shown in Figure 2.11.

Table 2.11: Refine Mesh

2D	Split	Displayed	Split
2D	split	displayed	split

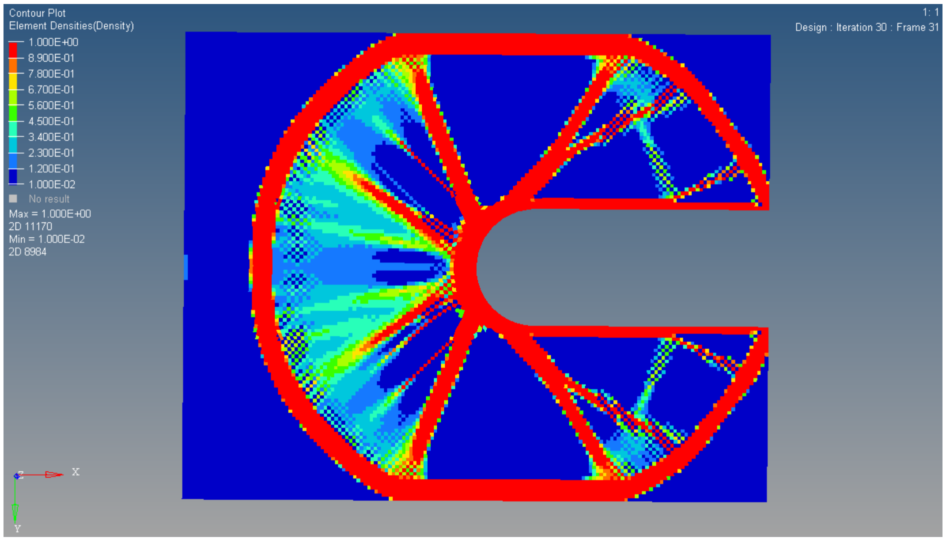
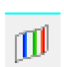
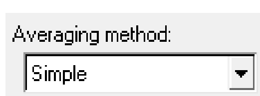


Figure 2.11: Refined Optimization Solution

Removing the lower densities to an effective range as shown in Figure 2.12 reveals the topology optimization in its finished form. Going to the slider below will reduce lower densities. Densities that are lower on the spectrum are less important to the structural integrity of the part and can be removed with the slider.

Table 2.12: Density Removal

Iso	Averaging method: Simple	Adjust Slider
		

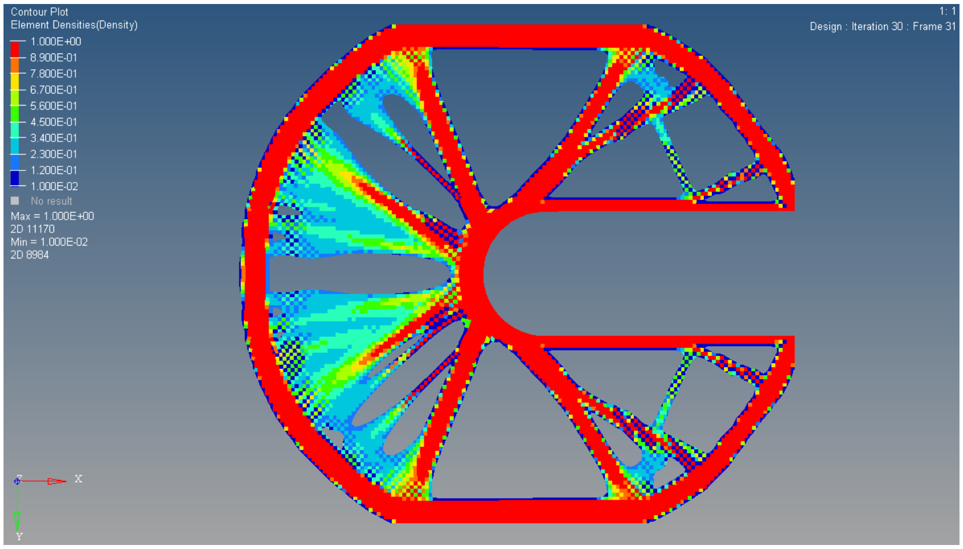


Figure 2.12: Optimized C-Clip

The resolution of the part can be further improved in a modeling program once the information gathered here is exported as an STL. Using this solution, a free-sizing optimization can be created afterwards to define the thickness of members shown in this optimization. Meaning, using this topology optimization will only show the location of the beam members. This structure still has a uniform thickness of 1 mm throughout. The beams may still be modified in the perpendicular (thinning or thickening the members) to apply the load more effectively. Free-sizing optimization, which will be covered next, is the following step in this process that will achieve this. The part exported through "export solver deck>STL>ok" reveals the 2D, final optimized shape in Figure 2.13.

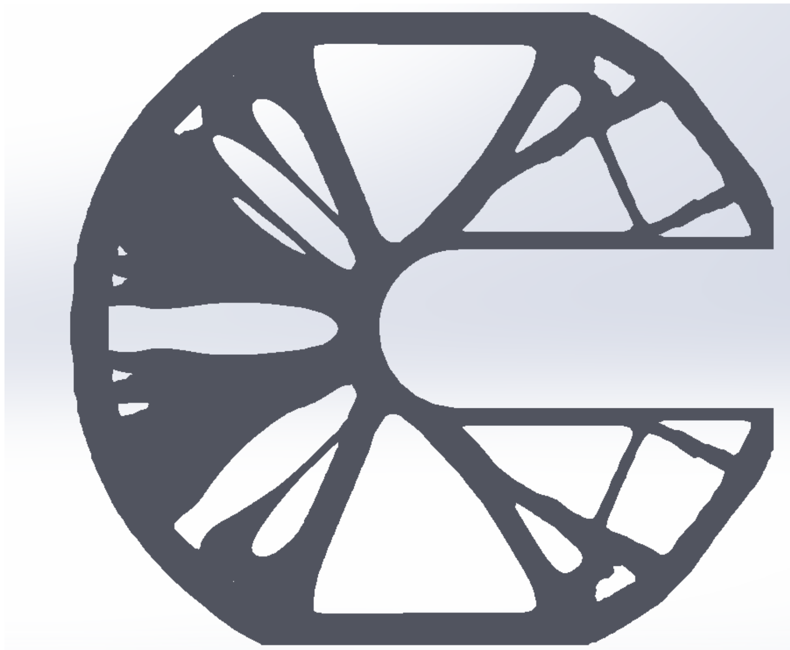


Figure 2.13: Completed 2D Stiffening Pattern in Microsoft 3D Builder

2.3 Analysis of Topology Optimization

Reviewing Figure 2.12 and 2.13, the final optimized part shows a moderately complex pattern of beam members derived from the parameters set. If placed flat, a mill would be able to machine this part with relative ease, depending on its size. This part is small and would require a machine with higher accuracy. One issue that many topology optimizations results have, including this one, is that there are members with a lack of continuity as well as inconsistencies and disconnects within the optimized mesh. For example, areas where the beam members connect with the outer rim of the C-Clip show inconsistencies as shown in Figure 2.14. If 3D printed, these inconsistencies in the mesh could cause stress concentrations and ultimately tearing of the part at these sections. Issues such as these show that there is still room for improvement in the software. Because of these inconsistencies, it is important to take a moment to analyze the results for consistency within the mesh and results. This shows that the state of topology optimization software right now is still more of a guide than an absolute solution. After a topology optimization, a stress analysis should be run to ensure that the stresses on the part are still within tolerance of the design.

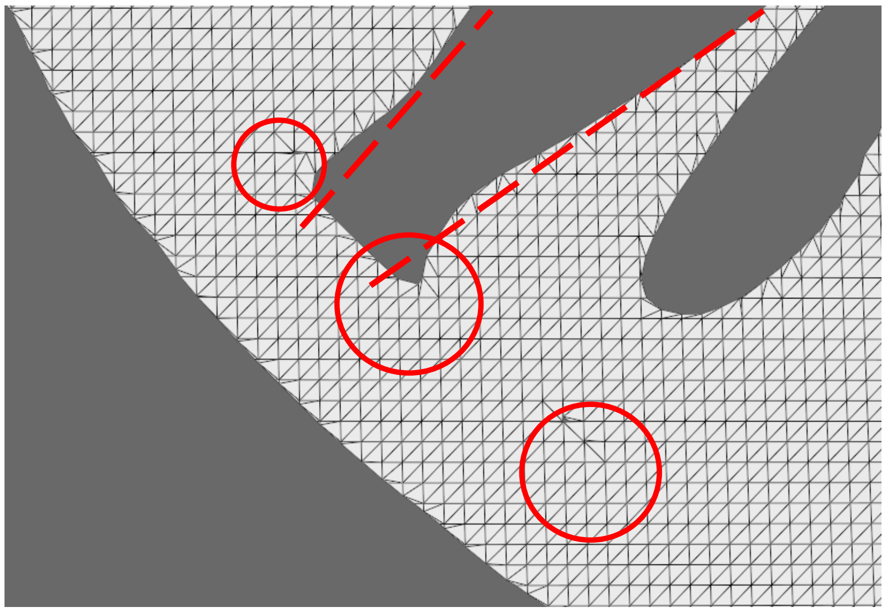


Figure 2.14: Optimization Disconnects Between Elements

It was also noticeable that when the resolution of the mesh was improved, a new set of beam members became visible. This is a strong proponent for mesh refinement, for without it, minor beam members that still carry an important amount of load would have been ignored. Although, while refining the mesh improved the resolution of the part, the amount of time required to optimize the part grew exponentially. It is therefor important to select the amount of time one is willing to invest in the level of accuracy that the program has to offer. However, with the student edition, this is a quantifiable limit that cannot be surpassed.

2.4 Application Within Aerospace

The 2018's 3D Printed Aircraft Fuselage Design team has applied topology optimization to its fuselage design with the objective of not only minimizing material usage as is shown in this C-Clip optimization, but also with the objective of maintaining the stiffness of the part. The design variable of that optimization is shown below.

The 2018 3D Printed Aircraft Design Guide includes reasoning on why these specific parameters were chosen. Performing this type of optimization on an aircraft component allows the user to see critical load lines caused by aileron, rudder, and elevators throughout the surface to reinforce them. In this instance, the fuselage is meant to be 3D printed with a single bead of thickness and requires a stiffening pattern throughout the inner surface that will carry this load. Figure 2.15 shows the design variable with parameters of Mindim 0.5 in and Maxdim of 1.0 in. This range means that the members are set to have thicknesses between 0.5 and 1.0. The properties of the ABS-M30 are applied to the fuselage with the property type to represent shell elements.

Name	Value
Solver Keyword	DTPL
Name	d_shell
ID	1
Include File	[Master Model]
Config	topology
Create	
Property Type	PSHELL
List Of Properties	1 Properties
Base Thickness	
Parameters	
Mindim	0.5
Stress Constraint	
Maxdim	1.0
Minumum Gap	☐
Fatigue Constraint	none
MeshType	☐

Figure 2.15: Fuselage Design Variable

In order to understand the process, a compilation of the evolution of the model results from the topology optimizations on the fuselage is discussed here. This history goes through the changes in application of the forces, the application of the objective, and the application of inward and outward pressure.

Figure 2.16 shows the simplified solution, which optimized all load cases by minimizing volume. This is the first successful run of the optimization function and yielded results that show clearly defined members outlined in solid red. If one were to do a sizing optimization from this topology optimization, one would find the thickest members closely following where the densities are highest in this image.

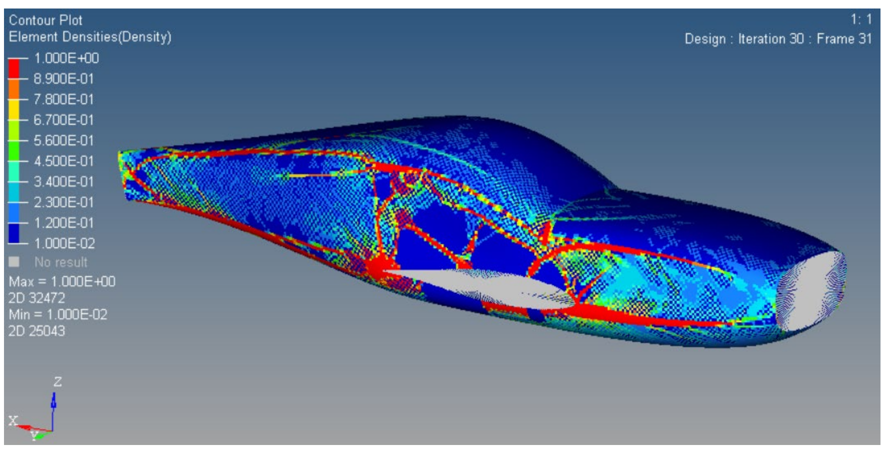


Figure 2.16: Volume Fraction Optimization (Iteration 1)

Through this, we learned that the initial decrease of the member size in the Design variable card showed an improvement in the member's solidity. This reduction in member size forced HyperWorks to create continuity throughout the model, whereas in the larger member sizing case, HyperWorks has issues with continuity. Another advantage is that a more ideal structure begins to form. When a larger member size is used, it is difficult to distinguish a stiffening structure. Part of that issue was apparent in Figure 2.17, where member thickness was overly large when moving from fractional volume to weighted compliance. However, a variable that was not taken into account was the minimum thickness of the members, which resulted in a poor quality optimization.

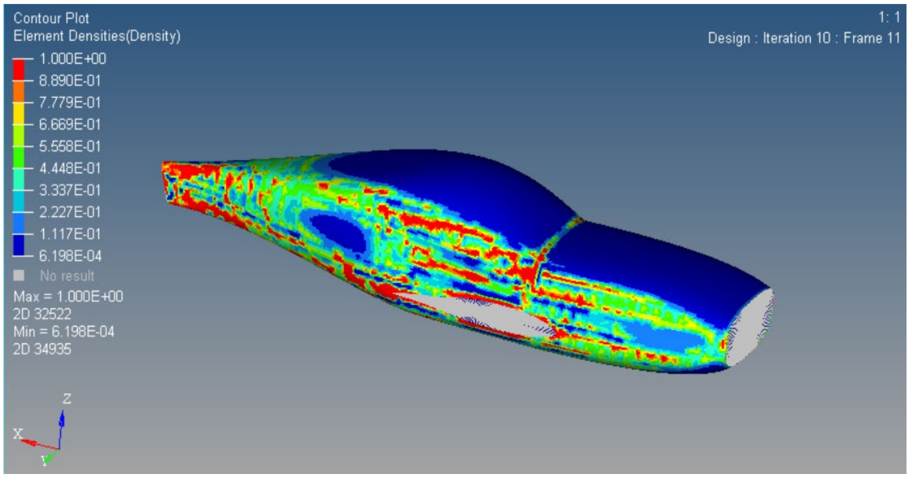


Figure 2.17: Disconnected Members (Iteration 2)

At this point it was brought to the optimization team's attention that the results of the optimization show a noticeable change in the structure, with thicker beams and a diminished hole at the back of the fuselage. In addition, no structure was generated at the top of the fuselage to ensure stiffness of the entire structure. It was then that the error was located in the minimum thickness parameter, and the optimization revealed more accurate, although excessive, member generations. Figure 2.18 displays these results.

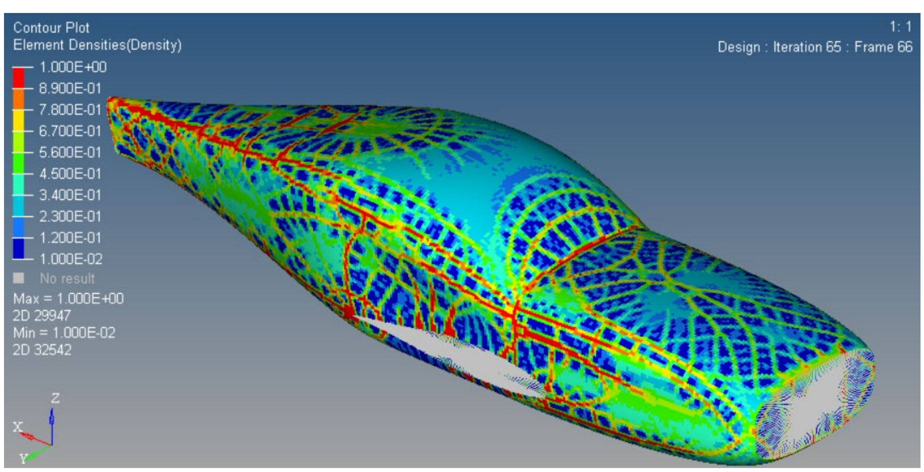


Figure 2.18: Effective Member Generation (Iteration 3)

While these members are indeed effectively transferring force, the member size is far too thin for proper modeling and 3D printing. In Figure 2.19, the final results of the topology optimization of the fuselage are shown. The model has a bounding box of approximately 28x6x6in. In relation to this bounding box, the members are still thin, but these thinner members will have different sized stiffeners that equate to these lower densities. It can be seen in red where the most critical sections are laid out. This is where hat stiffeners would be placed. Members in green represent relatively important locations where blade stiffeners would be placed.

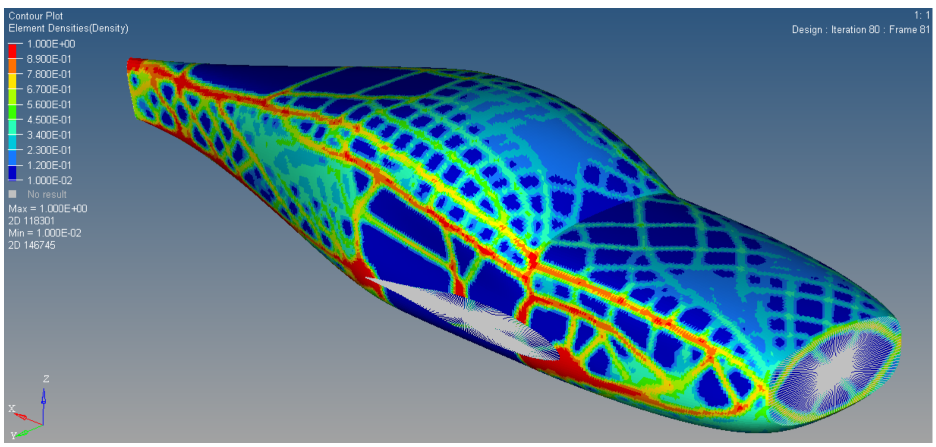


Figure 2.19: Topology Optimization of Fuselage (Iteration 4)

What is particularly unique about this application was the relative ease of it, since forces applied to the fuselage were made using RBE3 nodes to expedite the process. The key realization in this optimization was the understanding that the base thickness be set to zero for beam members to be clean and with effective connectivity. A stiffener pattern was later grafted on, which included patterns for blade, hat, and v-stiffeners followed by a static analysis of the resulting topology optimization. This can be viewed in detail in the 2018 Guide: *Optimization of the Internal Topology and Sizing for Thin-Walled Aircraft Structures – A Design and Production Guideline for Fused Deposition Modeling.*

CHAPTER 3

FREE-SIZING OPTIMIZATION

3.1 Free-Sizing Optimization Theory

Free-sizing optimization has the goal of finding the optimal thickness for any given part, be that in surfaces or solid models. Free sizing has the ability to create optimized depth and dimension to members of the topology optimization discussed previously, which only gave an understanding of the location and density of the beam members themselves. This shows the relative importance of the beams but does nothing for understanding the thickness of those beams. For that reason, showing the user how sizing optimization applies to beam members is vitally important. Therefore, this section will focus mainly on the results and what they impart to the user. The topology optimization of the C-Clip will be used in this example to further explain the optimization of members, followed by sizing optimization, which was a method that was integral in the optimization of the aircraft wing in 2018's Wing Optimization.

Name	Value
Solver Keyword	DSIZE
Name	d_shell
ID	1
Include File	[Master Model]
Config	free size
Create	
type:	PSHELL
List Of Properties	1 Properties
Minimum Thickness	
Maximum Thickness	5.0
Parameters	
Mindim	
Stress Constraint	
Fatigue Constraint	none

Figure 3.1: Free-Sizing Design Variable

The free-sizing optimization parameters are as shown in Figure 3.1. The free-sizing variable includes several parameters that deal with the thickness of a part normal to the direction of the surface. Minimum and maximum thickness dictate the range at which the optimization of the members in the normal direction can extend or remove its material. Minimum dimension can constrain the model further if there is a stress constrain or fatigue constraint included.

3.2 Guided Free-Sizing Optimization

This free-sizing optimization guide will show how to import, fix the imported mesh, assign the design variable, and interpret the solution. Here, the objective is to optimize a C-Clip for volume with a requirement that it withstand two forces of 100 N in in opposing vertical directions. There are two ways to go import free-sizing optimization results. These are explained in Section 3.2.1 and 3.2.2.

3.2.1 Re-Establishing the Model Environment

If OSSmooth does not work as suggested below, repeat steps 1-8 to regain the forces, material properties, etc, that were available in topology optimization.

3.2.2 Importing through OSSmooth

1. To import the topology optimization results, go to optimization, OSSmooth, then select the optimization model .fem file and select FEA reanalysis. The iso surface box should be checked and the threshold set at whatever density threshold from the previous optimization was most visibly effective. In this case, the threshold selected was 0.3. Then, select OSSmooth to import the file.

Table 3.1: Import Through OSSmooth

Optim.	OSSmooth	FEM model	FEA Analysis	ISO Surf	% Removal	OSSmooth
Optimization	OptiStruct OSSmooth	Select mode	FEA reanalysis	iso surface connection draw recov	threshold:	OSSmooth

Before the model is analyzed further, problem areas need to be addressed. This includes areas with small holes that need to be filled. If these small holes are left in the mesh, they will negatively affect the results of the next optimization, causing thicker areas where voids are present.

1. First the hole must be identified. An example of this would be the one such as the hole shown in Figure 3.2.

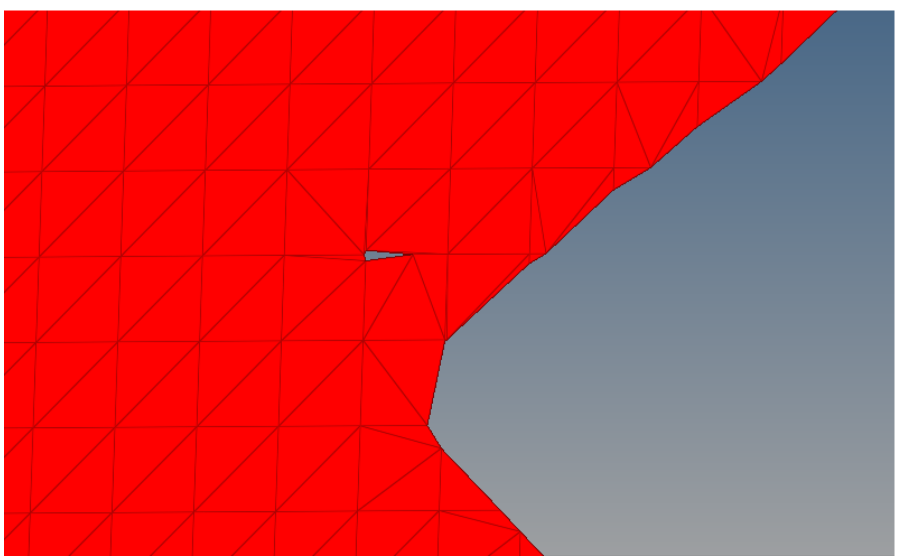


Figure 3.2: Mesh Defect

2. Once the defect is found, the area around it must be deleted to make way for the clean mesh that replaces it. This can be done by going to the tools and delete menu, and then selecting the elements around the void. Once these elements are

selected, they can be deleted. The holes should now display as shown in Figure 3.3.

Table 3.2: Delete Elements

Tool	Delete	Select Elements	Delete Entity
Tool	mask delete	elems	delete entity

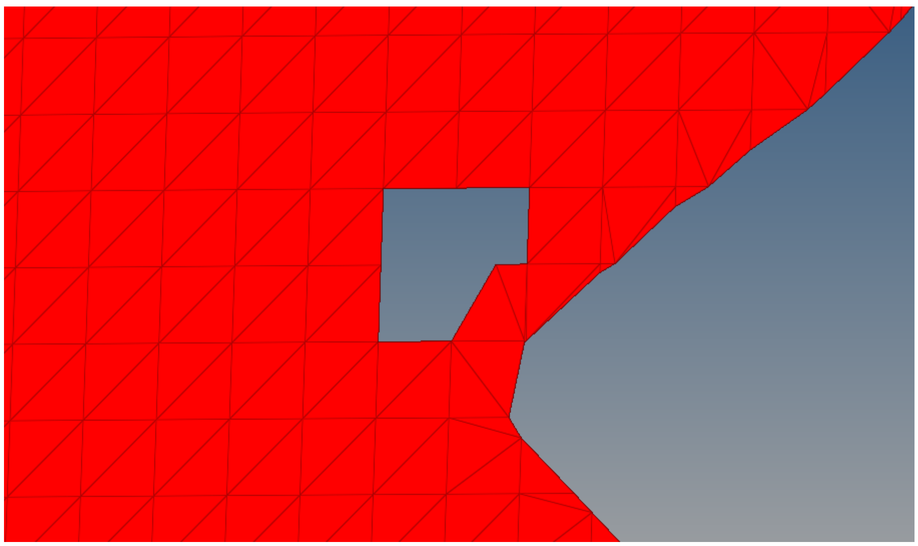


Figure 3.3: Removed Elements from Hole

3. Once the hole is cut, it needs to be filled. This can be accomplished through the mesh hole fill command. The results of this filling are shown in Figure 3.4.

Table 3.3: Fill Gap

Mesh	Hole/Gap Fill	Patch Fill	Edges	Proceed	Fill
Mesh	Boolean Operation Hole/Gap Fill	Fill Type Patch Fill	Select Entities	proceed	Fill

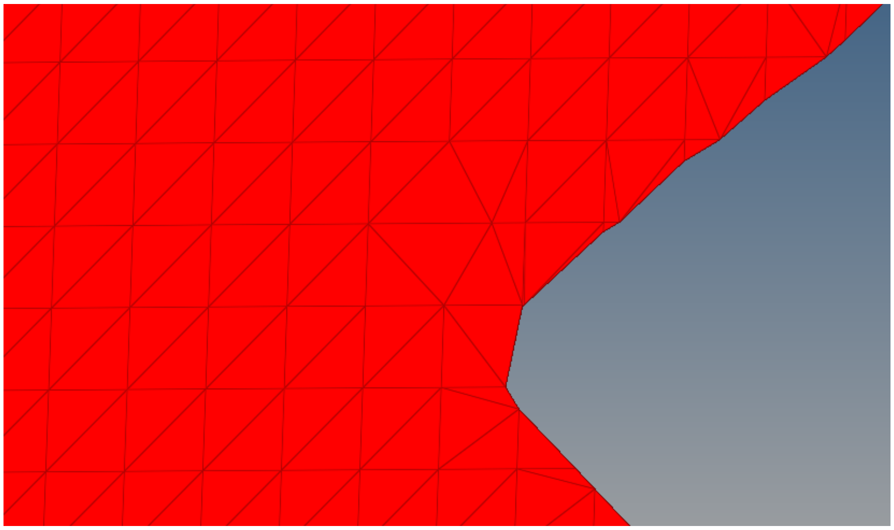
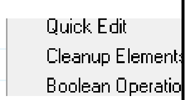
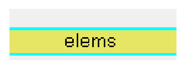


Figure 3.4: Re-Meshed Elements from Hole

4. Now that the holes have been filled, the mesh must be checked for issues.

Table 3.4: Clean Up Mesh Elements

Mesh	Clean Up Elements	Element Cleanup	Select Elements	Cleanup
				

5. If any elements remain that are causing issues, delete them again through Table 3.2 but instead of selecting the entities, select the "by id" method instead by clicking the elements pane.

3.2.3 Design Variable

To define the optimization as free-size optimization, redefine the optimization design by changing its configuration, redefining its property, and defining the thickness variable as shown in Figure 3.5.

Name	Value
Solver Keyword	DSIZE
Name	d_shell
ID	1
Include File	[Master Model]
Config	free size
⊟ Create	
type:	PSHELL
List Of Properties	1 Properties
Minimum Thickness	
Maximum Thickness	1.0
⊟ Parameters	

Figure 3.5: Design Variable Parameters

This time, use the optimized model and change the optimization type to Free Size optimization through the design variable card, keeping maximum thickness to 1 (mm).

3.2.4 Run the Optimization

To run the free-sizing optimization, use the method in Table 2.10. If successful, HyperWorks solver view will show a “Job complete” message similar to Figure 2.9 in a separate popup. The result is as shown in Figure 3.6.

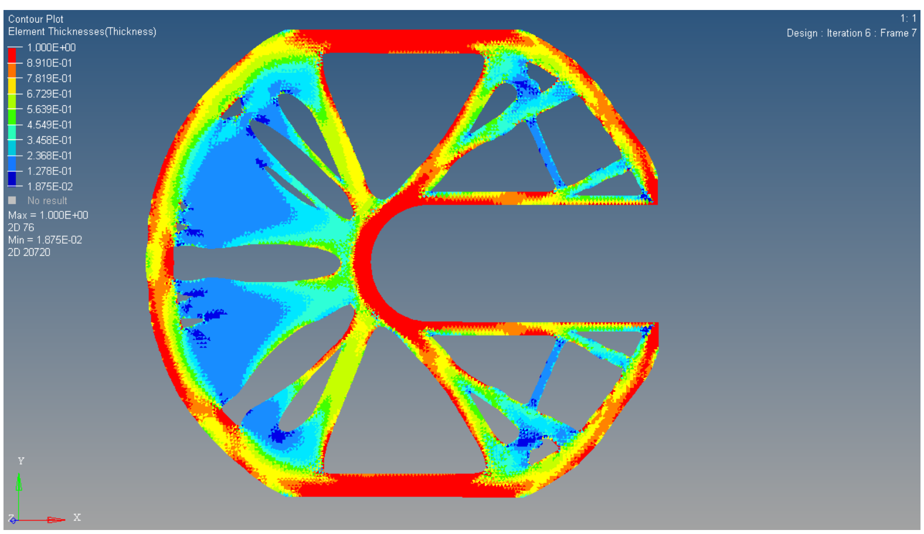


Figure 3.6: Free-Sizing Optimization Results

3.3 Analysis of Free-Sizing Optimization

Unlike topology optimization, which optimizes the density of the part, free-sizing optimization has optimized the thickness of each element in the mesh. Looking at the finished product, it is important to note that the finished optimization shows differing regions of thickness in key areas in bright red such as the hole at the center where the arms extend out as well as the beams on the top and bottom of the part. These key areas have a maximum beam thickness of 1mm, as defined by the design variable. Areas in blue are less essential to the part and are thinner. It is understandable that the members in red around the inner groove are thicker. This is to ensure that the part does not split in two down the middle, due to the forces that are pushing outwardly from the C-Clip's arms. It should be noted that essentially nowhere in the optimization does the program recommend that the thickness of the part go down to lower than 0.13 or 13% of totality. That means that every part does have some part to play. While areas in blue are not vitally important to the structural integrity of the part, it is likely that they are included in the optimization to relieve and distribute stress from critical beams in red to less critical beams labeled in yellow.

As interesting as the results are, this piece would not benefit from a free-sizing optimization considering how small it is, and there are a few reasons why. For one, the part is only 1 mm thick. This suggests that it is made of sheet metal. Machining something that is already so thin will cause defects in manufacturing. In addition, tooling for the part would not be possible save for temporarily gluing it to the machining bed. A possible manufacturing solution would be investment casting, which would be able to pick up the finer details at a much lower cost. The layers that are at their thinnest (in blue) are roughly 13%. This means that 0.13 mm of material would be left in that region. The machining would not be cost effective for such a thin piece. At this size a topology optimization alone would solve the problem without unnecessarily wasting time. However, larger parts with a similar necessity for reducing beam thickness using free-size optimization, such as the 3D printed fuselage, would benefit from such a process.

3.4 Application Within Aerospace

A free-sizing optimization has been performed on the fuselage to identify how tall the stiffening structure should be. The justification for optimization is the necessity of identifying the layer thickness of each location of the stiffener structures that will be placed on the fuselage. It is not enough to know only their location from the topology optimization. It is also important to understand the height need of the stiffeners to reduce the weight of the fuselage further. In Figure 3.7, the maximum thickness of the fuselage is labeled as 0.5 in., which is more than ample reinforcement for this design based on the strength of each layer of ABS. More information on the stiffening structures to be applied to the fuselage can be found in the 2018 Guide.

Name	Value
Solver Keyword	DSIZE
Name	d_shell
ID	1
Include File	[Master Model]
Config	free size
Create	
type:	PSHELL
List Of Properties	1 Properties
Minimum Thickness	
Maximum Thickness	0.5

Figure 3.7: Free-Size Parameter

Areas above 0.02 in, which are labeled in light blue require more reinforcement than the single bead thickness that the 3D printer can provide by itself. From Figure 3.8 and 3.9, the most critical areas are those at the corners of the fuselage, the back section, as well as members that connect both the top and bottom of the fuselage. Stiffening structures would be placed along these areas to provide this extra level of reinforcement that the optimization suggests. Notice that the blue section below the fuselage is dark blue. The material here is either at the threshold to 0.06 in, or below. These areas seem to require little, if any reinforcement, but in reality, these areas require a minimum thickness of at or above 0.2 in. This is due to the thin layer of material that should be ever present across the entire fuselage for aerodynamic and protective shielding of passengers and components.

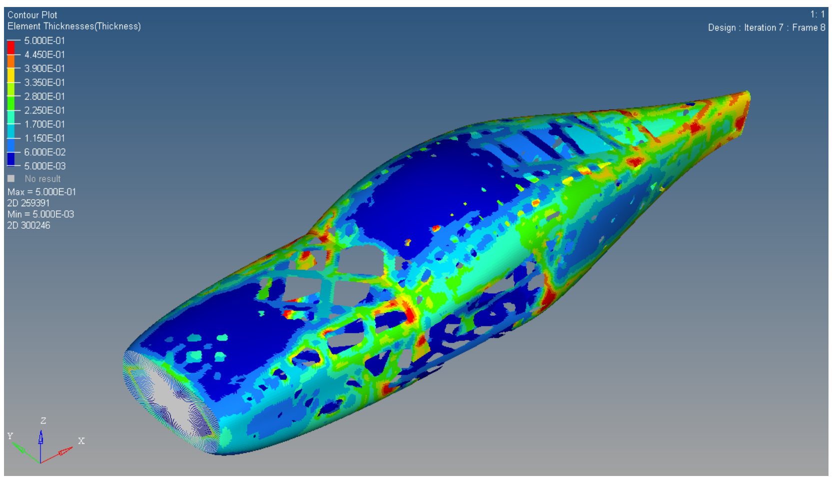


Figure 3.8: Free-Sizing Optimization (Top View)

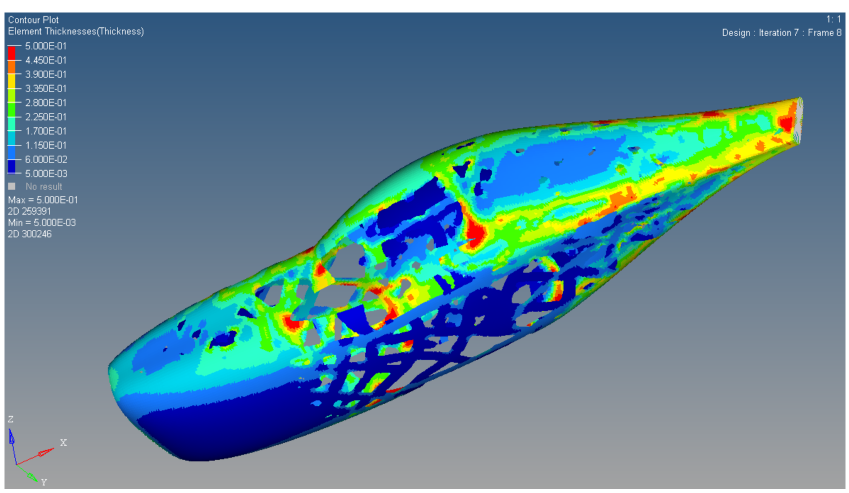


Figure 3.9: Free-Sizing Optimization (Bottom View)

As this is an approximation of the properties of the ABS M-30 material, it is hard to say whether there will be issues in the material. 3D printed material represents a limitation in this manner because the material properties are often times a product of the printer's ability to lay down material with a uniform rate, producing uniform material properties without creating material discontinuities and defects. Extrusion printing with thermoplastics especially lacks the ability to promise a perfect print, and thus these results need to be considered. The thickness of the material may need to be increased to counteract this effect. Stiffeners used in the build of the fuselage are also hollow in nature, and this makes it difficult to create a true picture of what an effective shape optimization would look like. In the future, the next design team could create a more effective solution by modeling stiffeners within HyperWorks and creating a shape optimization based on that model. Whether it would be worth the effort to shave a few lines of material off the final model, however, is debatable.

Due to time constraints, this optimization was not used on the fuselage. The information gathered here will help the next MAE 4188 Senior Design team perform a more effective solution with varied stiffener sizes to further minimize material usage.

CHAPTER 4

TOPOGRAPHY OPTIMIZATION

4.1 Topography Optimization Theory

Topography optimization is a type of shape optimization that bends thin structures into a shape that is more efficient for the forces applied to it. Unlike topology and free-sizing optimization, the thickness and density of the part cannot change. For example, in sheet metal fabrication, topography optimization is used in conjunction with some sort of press to deform the sheet into a shape that most effectively receives the force and expels it through the part's constraints. It should be noted that topography optimization only applies to surface models. Therefore, a new force must be applied to the previous C-Clip model that is out of plane in order to explain the effects of topography optimization.

The topography optimization card consists of a basic interface (Figure 4.1), where topography can be disabled or enabled.

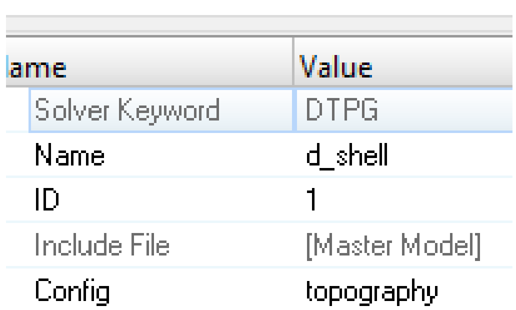


Figure 4.1: Topography Variable

However, topography cannot run on only this as it must have certain restrictions that are not plainly listed by the card. This seems to be a flaw in the user interface. A design space and bead definition must be defined before the topography optimization can begin. The design space of an optimization consists of the area that is under optimization, or rather the area that should be shaped and distorted to suit the optimization's requirements. The optimization also requires a bead definition. This includes information on the minimum width of the beads of the model as well as the maximum height that the material may be drawn to. Applying these constraints is explained in Section 4.2.6.

4.2 Guided Topography Optimization

This topography optimization guide will show how to establish the model environment, assign a new force, assign new constraints, adjust the responses, creating a design objective reference, assign the design variable, and interpreting the solution. Here, the objective is to minimize the deflection of the nodes where forces are applied. The added force will have a maximum deflection of 4mm, while the forces will allow to deflect by only 0.07mm. The guide begins where the topology optimization ends, in the main HM file.

4.2.1 Re-Establishing the Model Environment

1. Open the topology optimization HM master file.
2. Remove the constraints through deletion as explained in Section 3.2.2.

4.2.2 Assigning a New Force

1. Create a force of 100 N. on the left end of the bracket, at its vertical center as explained in Section 2.2.5.
2. The applied force should be as shown in Figure 4.2.

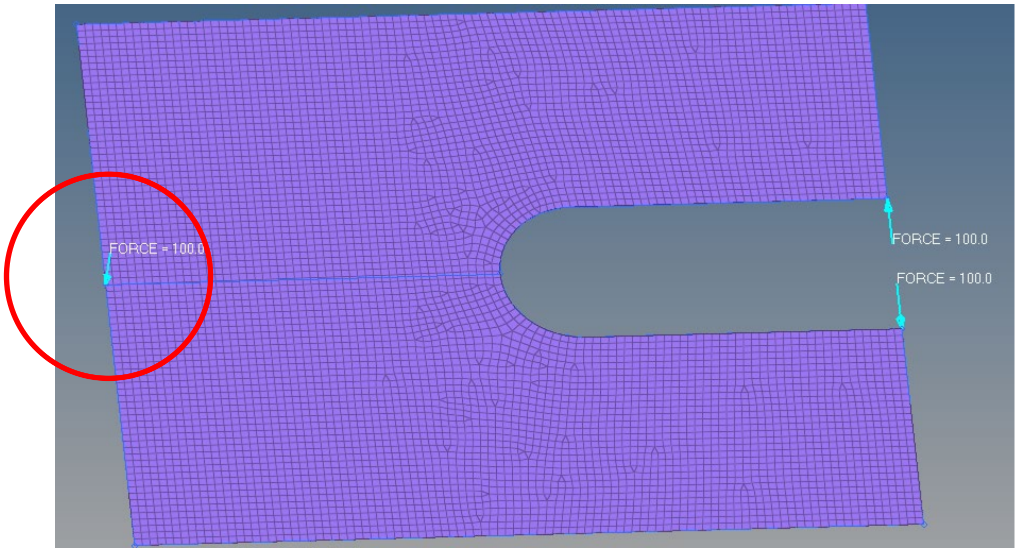


Figure 4.2: Addition of a Force

4.2.3 Assigning New Load Constraints

1. As in section 2.2.5, assign new load constraints to the now empty active load collector "constraints" as shown in Figure 4.3.

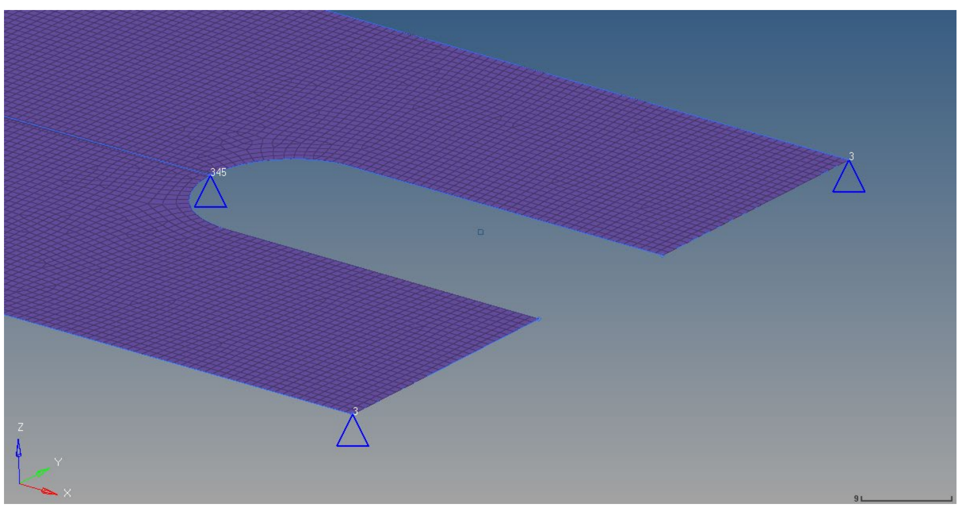


Figure 4.3: Constraints Application

4.2.4 Adjusting the Responses

The response of the new forces of 100 N should be accounted for.

1. Responses may be created using the guide in Table 2.5 with 4 responses with parameters as dictated below.
    a. Upperdis (already in model).
    b. Lowerdis (already in model).
    c. Verticaldis: 1 node, with dof3 for the 100 N. force on the left of the part.
    d. Total_disp: 3 nodes, with total displacement.

4.2.5 Creating an Optimization Constraint

Constraints must be adjusted to minimize the deflection of the part's arms and the left hand side of the body.

1. Constraints may be created using the guide in Table 2.6 with 4 responses with parameters as dictated below.
2. C_upper: (already in model).
3. C_lower: (already in model).
4. C_vertical: bounds of -4.0 to 4.0, response set as *verticaldis*, with one load step applied.

4.2.6 Adjusting a Design Objective Reference

1. To adjust a design objective reference to check all load steps, edit the initial objective reference and input input Max_Disp as the name, with a check mark in positive response and a 1.0 in the entry area.
2. Select Total_Disp as the response and all for the load steps.
3. If no design objective has been created yet, view Table 5.1 for instruction.

4.2.7 Design Variable

The design variable in topography optimization should be defined as shown below.

1. As in previous optimizations, go the design variable card and change its configuration ("config") and change it to topography.
2. Navigate to edit the full topography optimization card by going to the optimization pane to make changes in the topography design variable card. In bead parameters, set the minimum width to 2, the draw angle to 60, and the draw height to 10. The draw direction should be normal to the elements and the boundary skip should be "load & spc", which are the same loading parameters currently on the C-Clip.

Table 4.1: Edit Bead Parameters

Analysis	Optimization	Topograph y	Bead Params
Analysis	optimization	topography	bead params

Selecting a minumum width as 2 inches which is 2x the nodal distance between elements. The recommended value is usually between 1.5x and 2x the size of a node. The draw angle defines the angle at which the optimization raises/lowers the material. Lastly, draw height defines the height at which the program can raise/lower the material. A buffer zone is added to ensure that elements outside the zone remain undisturbed by the optimization.

3. In the bounds, change the upper and lower bounds to 1.0 and 0.0 respectively.

   Click update.

4. In the update panel, select the relevant property and click update once more.

4.2.8 Run the Optimization

To run the free-sizing optimization, use the method in Table 2.10. If successful, HyperWorks solver view will show a “Job complete” message similar to Figure 2.9 in a separate popup. The result is as shown in Figure 4.4.

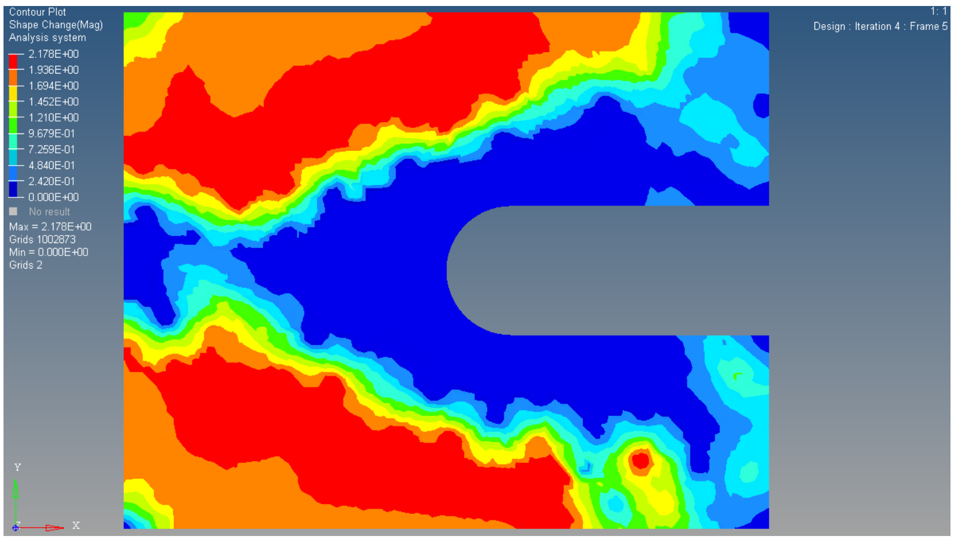


Figure 4.4: Shape Change Optimization Results

For a better understanding of the change that the shape underwent, Figure 4.5 is supplied.

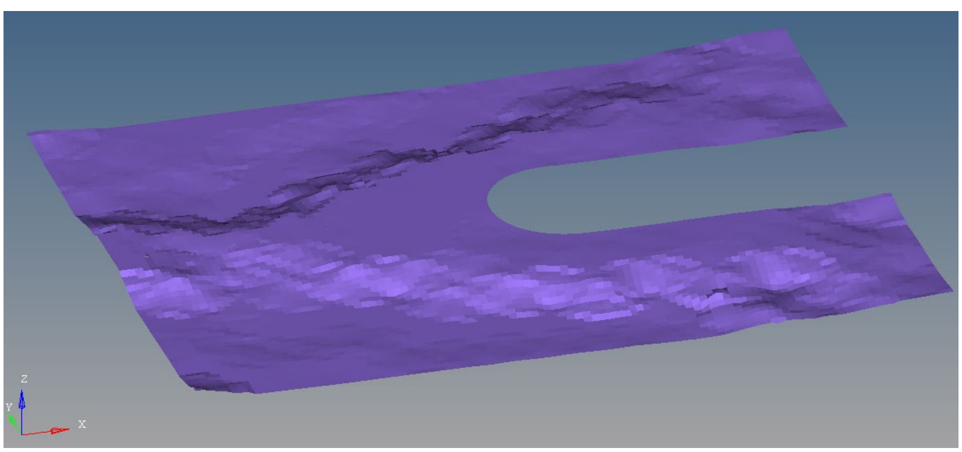


Figure 4.5: Visual Representation of Topography Optimization

4.3 Analysis of Topography Optimization

Unlike sizing and topology optimizations, this optimization mainly deals with shape change, as elevation is lowered. As the color goes from dark blue to bright red, the height change in the part increases. Notice the symmetry of the optimization. This is due to the part's equal distribution of forces across both the top and bottom sections of the part. It is important to note; however, that the optimization symmetry is not perfect. The irregularity of the mesh and the optimization's understanding of the part is not ideal in this case. To improve the results, HyperWorks has a symmetry tool.

Reviewing the topography optimization, it is noticeable that the sheet has deformed in an understandable way. The location of the vertical force has formed a ridge to counteract this deflection. Because of this, a deflection that would have resulted in 7.5mm now only results in a 4mm deflection. In stark contrast to this change, the arms of the C-Clip have remained relatively unchanged in elevation. This is due to the nature of the forces that are acting on it. Since these forces are in-plane, there is not much that the optimization can do to improve this area other than maintaining relative flatness and transferring some of the load from the vertically applied force by gradually returning the deep ridges on the left of the part to a flattened sheet.

4.4 Application Within Aerospace

A free-shape optimization was not performed on the fuselage due to the fact that the 2018 Design team's material was not any type of deformable material. We are strictly limited to 3D printed ABS M-30. However, this type of optimization could be used for thin aerostructure components inside the fuselage to secure electrical components, boards, etc.

CHAPTER 5

FREE-SHAPE OPTIMIZATION

5.1 Free-Shape Optimization Theory

Free-shape optimization modifies the shape of the part to fit certain parameters. For surface objects, it is much less defined of an option than free-size optimization is, where you can choose the member thickness. In free-shape optimization, a work area can be selected and the shape of this area altered. Free-shape optimization optimizes the outer boundary of a structure given some pre-defined objective and constraints.

As with the topography optimization card, the free-shape optimization does not list anything of value see (Figure 5.1). Most of these important parameters are tucked away in submenus that must be discovered and assessed. The most basic of parameters is the design space that must be selected.

Name	Value
Solver Keyword	DSHAPE
Name	d_shell_shape
ID	2
Include File	[Master Model]
Config	free shape

Figure 5.1: Free-Shape Optimization

Application of the Free-Shape optimization design variable is explained in Section 5.2.4.

5.2 Guided Free-Shape Optimization

This free-shape optimization guide will show how to re-establish the model environment, assign a new force, assign a static displacement response, minimize nodal displacement, creating a design objective reference, assign the design variable, and interpret the solution. Here, the objective is to minimize the deflection of the nodes where forces are applied. The added force will have a maximum deflection of 4mm, while the forces will allow to deflect by only 0.05mm. The guide begins where the topology optimization ends, in the main HM file.

5.2.1 Re-Establishing the Model Environment

Take the completed topology optimization environment HM file and copy it into another folder. Here, we will do a free-shape optimization on the model to further reduce the displacement of the nodes at the ends of the clip by changing only two beam members.

5.2.2 Applying a Static Displacement Response

The volumefrac response only works for topology and sizing optimizations, so it must be removed and replaced with another objective. Replace it with a static displacement.

1. Replace the volumefrac by changing the response type to static displacement.
2. Rename the item to “total_displacement”.
3. Select the two nodes where the forces are reacting on the C-Clip.
4. Change the constraint type to total displacement.
5. Create a Design Objective Reference that changes all displacement values to positive so that the objective may minimize them. During creation, apply the reference to all load steps.

Table 5.1: Static Displacement Response

Analysis	Optimization	Obj Reference	Positive Reference (LS: all)	Create
Analysis	optimization	obj reference	neg reference = -1.0 pos reference = 1.0	create

Change the objective format to an objective type of Minmax with a “List of Dobjrefs” that points to the Max_Disp design objective reference.

5.2.3 Increasing Displacement Constraints

The displacement on the C-Clip needs to be further constrained than before.

1. Change c_upper and c_lower constraint levels from 0.07 and -0.07 to 0.05 and -0.05, respectively.

5.2.4 Apply the Free-Shape Design Variable

Next, the design variable must be altered. However, HyperWorks does not seem to realize that the design variable for free-shape optimization needs to select nodes from the card when the free-shape is selected. Because of this, the old design variable must be delete, and a new design variable created.

1. Delete the old design variable and create a new design variable using the free shape optimization menu.

Table 5.2: Create Shape Optimization Variable

Analysis	Optimization	Free Shape	Create	Select Nodes	Create
Analysis	optimization	free shape	create	nodes	create

2. The nodes can be selected by holding down shift and clicking the right button once to select the circle tool. Then, hold shift down and click and drag a circle over the two beam member regions. The selection should look like that in Figure 5.2.

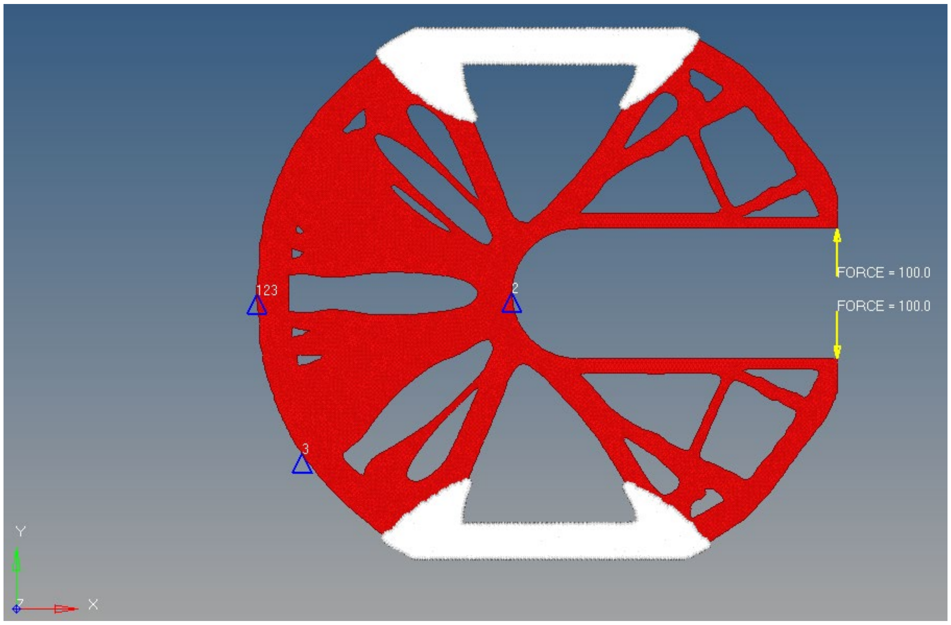


Figure 5.2: Nodal Selection of Beam Members

5.2.5 Run the Optimization

To run the free-shape optimization, use the method in Table 2.10. If successful, HyperWorks solver view will show a "Job complete" message similar to Figure 2.9 in a separate popup. The final optimized shape of the selected beams is shown in Figure 5.3.

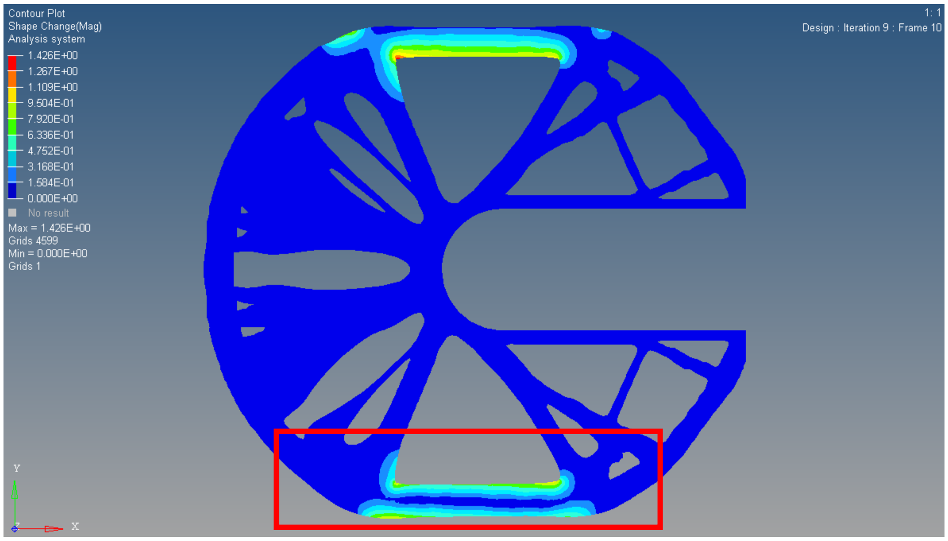


Figure 5.3: Free-Shape Optimization Over a Design Space

Figure 5.4 expands on Figure 5.3 to reveal the direction of movement of the mesh elements to counteract the new constraint on displacement.

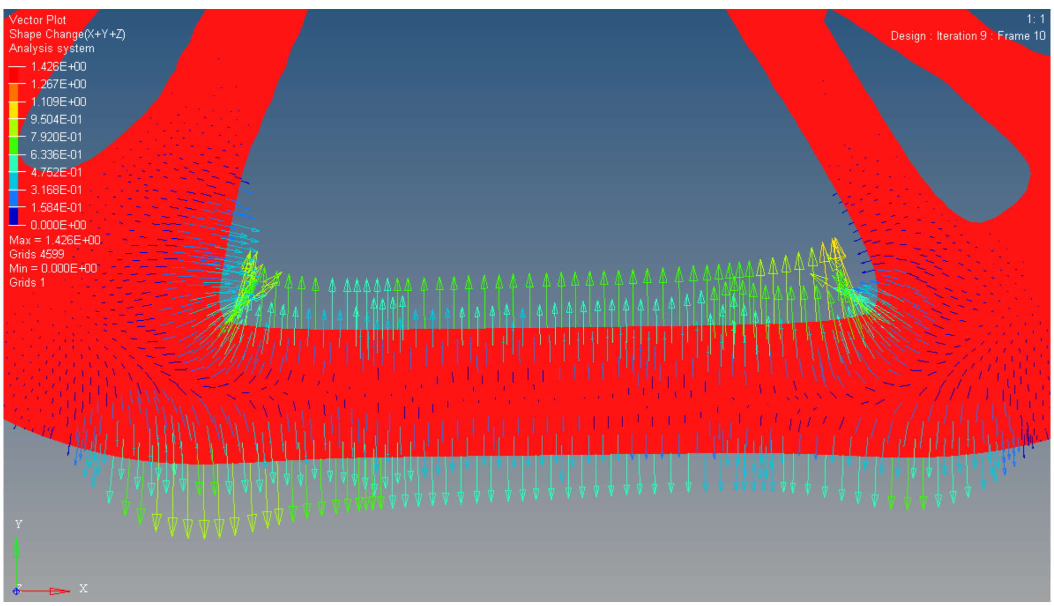


Figure 5.4: Free-Sizing Optimization, Vector Plot of Mesh Migration

5.3 Analysis of Free-Sizing Optimization

In the free-sizing optimization, it was found that one of the most important areas for material to be deposited in was the ring around the C-Clip. This area carried a large amount of load, and therefore the displacement of the C-Clip would be lessened if more material were to be applied in that area. Using this reasoning, the displacement was able to be reduced a small fraction from 0.07mm to 0.05mm by increasing material in only these two members. It should be noted that adding material to other beams besides those labeled in red in the free-sizing optimization and the area in the center of the C-Clip would not result in any meaningful reduction in deformation. Optimizing for beam members, other than these, could result in the optimization trying to enlargen the beam excessively. The software could attempt to grow the beam to the point of eliminating it in an attempted to meet the constraints of the design. This would not actually pose a design issue, but more of a material usage issue. In essence, it would not be efficient. By using both beams at the top and bottom reduces the amount of material needed to reduce the static displacements from the forces due to the fact that these areas are more pivotal regions in the design. Notice in Figure 5.3 that the free-shape change does not affect the overall build of the part, but rather increases/decreases element size to minimize the displacement of the nodes at each tip.

Looking closer at the optimization, there are areas for improvement. For example, corner areas where the optimization could not minimize the element further such as shown in Figure 5.5 create geometric problems that must be eliminated after the optimization. If one where to directly export this model to another FEM analysis software such as ANSYS, or 3D print this model, there would be a serious issue with localized stresses in this area. These issues could possibly result in errors or failures in the part depending on its intended use.

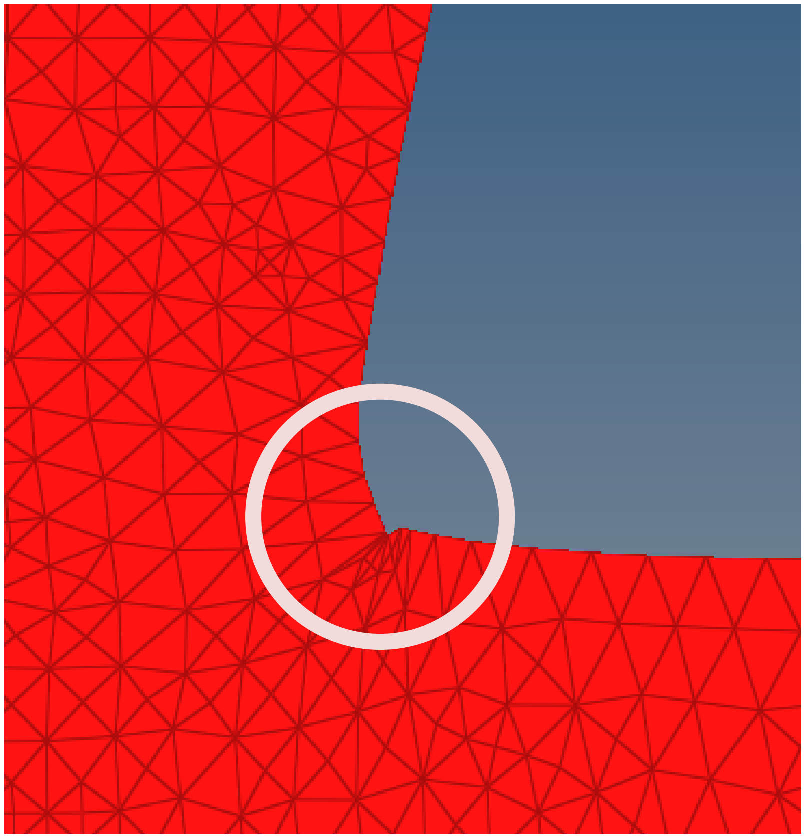


Figure 5.5: Sharp Disconnect in Mesh

5.4 Application in Aerospace

A free-shape optimization was not performed on the fuselage. This is strictly because the fuselage is not using the in-plane width of the beam members in its assessment of stiffener size. For example, the blade stiffener on the fuselage, as shown in Figure 5.6, would not benefit from having its width reduced any further. This is also the majority of stiffening structures being used in the fuselage.

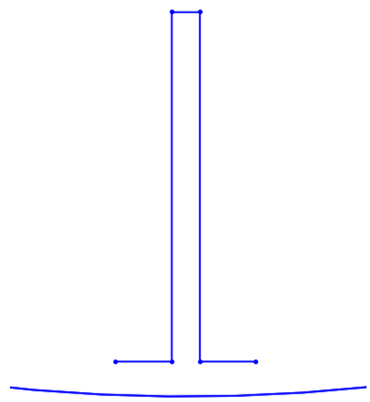


Figure 5.6: Blade Stiffener

CHAPTER 6

CONCLUSION

While all four optimizations have their own merit, only topology and sizing optimization are readily applicable to the senior design project. In the 2018 Senior Design Guide, the objective was to manufacture with 3D printing in mind. Topography optimization was not used due to the fact that it is only applied in circumstances such as forming or swaging techniques. In addition, shape optimization was not utilized on the stiffening structure because the stiffening structure's width, for example the blade stiffener, cannot be cut down any further. In addition, the design guide summarized four pivotal optimization techniques: Topology optimization, Sizing optimization, Shape optimization, and Topography optimization. The guide included how to navigate through its setup, with insight as to how each card functions within the model. The paper also discussed the pros and cons of material optimization. This included advantages such as saving on material costs, cutting down on the weight of an object, reducing time spent redesigning the optimization, maintaining stiffness, and optimizing for many more such responses available in the objectives selection. Also included were the downsides of material optimization, including the risk of optimizing too early in the design phase, the cost to manufacture an optimized part, the time and resources required to optimize that part, and the lack of wiggle room with regards to defects are among the few reasons. In summation, optimization is a powerful tool. If used correctly, the techniques shown in this design guide will help others understand the method involved in optimizing a part for structural efficiency.

REFERENCES

Bendsoe, Martin Philip, and Ole Sigmund. “Topology Optimization: Theory, Methods, and Applications.” *Google*, Spinger-Verlag Berlin, 2003.

Bhate, Dhruv. “Additive Manufacturing - Back to the Future!” *Padtinc*, WordPress, 28 Jan. 2016, www.padtinc.com/blog/additive-mfg/additive-manufacturing-back-to-the-future.

Chandole, S. K., et al. “STRUCTURAL ANALYSIS OF STEERING YOKE OF AN AUTOMOBILE FOR WITHSTANDING TORSION/ SHEAR LOADS.” *East Journals*, IJRET, 1 Mar. 2014.

McCullough, Shawn, et al. Fused Deposition Modeling of Thin-Walled Aircraft Structures. 3rd ed., UTA, 2017, pp. 1–111, Fused Deposition Modeling of Thin-Walled Aircraft Structures.

Risberg, Martin. “Topology Optimization of Castings.” *Altair ATC*, SWERA, 3 Nov. 2009.

BIOGRAPHICAL INFORMATION

Alexandra Kessler is a Mechanical Engineering student at The University of Arlington (UTA) in her senior year. She hopes to continue her studies as a master's student after graduation from UTA and a two-year position at Textron. As a Dean's List student, she ranks within the top 183 of 4000 students in the Engineering College and has worked hard to excel in her studies and service to her college and community, being both a member of Alpha Phi Omega (APO), a national service fraternity, the Society of Women Engineers (SWE), and the American Society of Mechanical Engineers (ASME). She has worked in various enterprises including Bell Helicopter, AECOM, Terpel Colombia, and Amazon.