Commercial Bank & Business Equipment (CBBE):

Bench v2 Optimization

Team Name: Kennesaw Consulting Company (KCC)

Authors
Garrett Smith - *Project Manager, Process Engineer*
Ernie Rivera - *Project Coordinator, Systems Engineer*
Austin Hester - *Technical Expert, Operations Research Engineer*
Jake Santa Cruz - *Data Analyst, Quality Engineer*

Advisors
Dr. Adeel Khalid - *Associate Professor of Industrial and Systems Engineering, KSU*
Will Mack - *VP of Special Projects, CBBE*
Dennis Batin - *Director of Engineering, CBBE*

Kennesaw State University
Department of Industrial and Systems Engineering

Executive Summary

The client is a furniture production company located in Woodstock, GA. One of their products is a bench used for café style settings in a major United States bank chain. The client defined their requirements with a prioritized list of wants, needs, and problems that they encouraged us to accomplish. The number one item on their list is that the benches are heavy and awkward to turn, flip over, and otherwise manufacture and maneuver, a potential safety hazard for workers.

After reviewing several options for improving the company's manufacturing process, Finite Element Analysis [FEA] was determined as the team's highest impact method for assisting the company. FEA tests a product without costly machinery or numerous sample products, "[allowing] the removal of the indefiniteness before the manufacturing and making the decisions related to manufacturing in a more healthy and economical manner" (Koç, et al., 2011).

The AutoCAD model of the bench was acquired from the client. Slight alterations were then made to the model to make it capable of undergoing stress testing, including drilling holes and adding bolts and screws. Then, eight alternate models were made, ranging from using plywood instead of Birch wood to less supports and larger holes in the interior. Stress was applied in three evolving ways as the team strived for a realistic method: one point force in the

center, two point forces on top of the interior support, and two pressures applied to a 144 square inch square.

One important note is that “the intention of performing FEA is not to eliminate the real tests but rather to reduce the time for a product to pass through the process” (Rundgren and Wörmke, 2011). Using Safety Factor as the primary metric for determining if a model is viable, the team came to the conclusion that either the baseline model or the baseline interior replaced with Plywood were the most viable options.

Table of Contents

List of Figures

List of Tables

Chapter 1: Introduction

The client is a furniture production company located in Woodstock, GA that focuses on furniture for use in bank branches such as teller counters, check desks, benches, and other types of commercial furniture. Currently, the client has begun production of the second version of a bench for use as seating in these bank branches as can be seen in **Figures 1 & 2**. However, upon deployment of the new product, the client began to notice a few problems arise and that there were many ways in which the process could be improved. After going through many different iterations of the product itself, the client decided to seek Kennesaw Consulting Company's (KCC) help in order to uncover and recommend improvements that can be implemented within production forward to shipment and assembly.

Figure 1 - Bench Front

Figure 2 - Bench Back

1.1 - Statement of the Problem

After visiting the client's warehouse and speaking with the staff, the KCC team was able to get a better idea of the wants and needs of the client, as well as identify many key issues that needed to be addressed. First, the benches are extremely heavy and awkward to manipulate. This is not only a safety hazard for the workers, but it also creates a disparity between male and female staff, while also increasing downtime when an additional worker must be called to help. Next, there is potential for improvement within the materials and machines themselves. For example, much of the spray glue and foam that is being used for the padding goes to waste and takes a long time to apply and to dry. Additionally, time on the CNC router machine, which is used for making precise cuts, is expensive and there is currently a lot of downtime. Another large issue the company is having does not occur until the very end of the process, even after the product has been delivered. Since the benches are shipped to a bank branch that is likely in the process of being built, it would be received by a general contractor on the site. Since this person has likely never seen or handled the product before, they are generally unfamiliar with how to uncrate and assemble the product so that the end result is undamaged and clean. As a result, the client has received several returns, which often lead to even more damaged products when they are repackaged incorrectly and shipped back. Because of this, the client has had to spend money and resources to send experienced employees to either assemble the product on-site or facilitate the return. It is the goal of the KCC team to solve these three main issues in order to create a lasting positive impact on the process.

1.2 - System Overview & System Block Diagram

The client provided us with a description of their product as follows: The upholstered 2-person bench is cut from plywood on a CNC router. The frame is assembled by hand using screws to prevent creaking. The bench base and back are sprayed with glue, wrapped in high-density foam, and covered with upholstery material. The base and the back are combined using a metal bracket purchased from an outside vendor. The baseplate and feet are attached and the completed bench, along with another is placed in a crate for shipping.

1.3 - Design Requirements & Specifications

The product is a piece of furniture for use as seating in new or remodelled bank branches. Specifically, the bench is used for “cafe” setups, which are designed to be more casual than traditionally visiting a bank. On-site, the benches face each other to encourage conversation between the consumer and the bank teller. This can be seen in **Figure 3**.

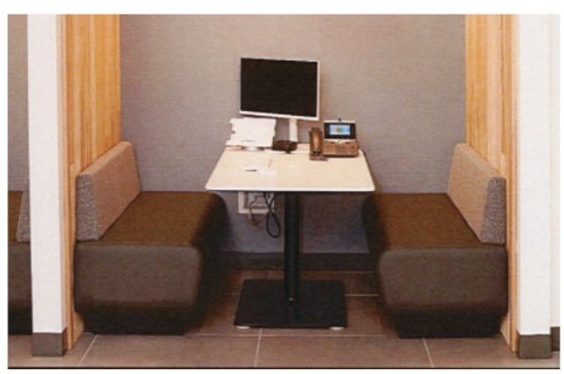


Figure 3 - Bench Model

The product is assembled in batches of 12 on site. The precise design dimensions of the bench are confidential, however, the CAD model of the bench with the specifics was provided to the team after they signed a Non-Disclosure Agreement. For the purpose of confidentiality, many aspects of the design including the name and other elements have been modified or left out completely. The company prefers that KCC not make drastic changes to the design itself, if any, especially when it comes to the appearance. Currently, the client suffers from a number of production issues that were mentioned in the problem statement. As such, KCC has performed studies and tests based off of three different solution approaches. First, the team considered implementing a type of lifting technology to reduce the cycle time for each bench as well as reduce the strain to employees. Also, the team has considered investigating the materials themselves that are currently being used. For instance the team would like to implement sticky back foam to save time and money in the padding process. Additionally, if weight can be reduced in these materials, it would help solve both of these issues. Finally, the team has considered implementing an improved packaging system to save money on shipping, as well as a new and improved instruction manual to reduce the number of returns that need to be processed.

1.4 - Design Concepts & Trade Study Items

Since the product is manufactured and marketed under a Non-Disclosure Agreement with the client, our team is not allowed to directly show dimensioned pictures of the design of the finished product. The client has provided the team with a number of confidential documents to help support optimization efforts. These documents range from blueprints of the product itself to assembly manuals to be used by the general contractor receiving the product.

1.5 - Verification Approach, Analysis and Simulation Tests

Our team is interested in performing a time study of the assembly, glue, foam, and upholstering portions of the manufacturing process and determining which portions of the assembly can be shortened most significantly (i.e. transport between stations, adhering foam, upholstering).

1.6 - Minimum Success Criteria

As stated by the client: “ideally the project will uncover and recommend improvements that can be implemented in production.” In the immediate meeting, we were given a lot of information as to possible areas of improvement within the scope of the Bench v2 program. The client did not put pressure on us to adhere to any one particular of those improvements and merely provided us with a list of wants in order of prioritization.

Chapter 2: Literature Review

2.1 - Material Flow

Material flow through a facility is a major source of increased cost and time for a company. According to Zhao and Wallace (2014) “an efficient layout of facilities can reduce operational costs and contribute to overall production efficiency”. In an effort to increase efficiency, “[Material flow] analysis plays a crucial role in detecting potential bottlenecks in production and in searching for a more efficient process layout, characterized by the reduced negative impact of these bottlenecks on system performance” (Siemiatkowski, et al., 2019). Studies have shown that “A significant proportion of overall production costs, about 20%–50%, can be attributed to material handling, since material handling is involved in almost all manufacturing activities from the arrival of raw materials to the packaging of finished products” (Zhao and Wallace, 2014). “Facility layout design (FLD) is a crucial task in redesigning, expanding, or designing the manufacturing systems, e.g., flow shop systems” and can be an expensive task for companies (Azadeh, et al.). In their paper on the optimization of facility layout, Azadeh, et al. described the most common objectives of layout problems to be:

> Minimization of the transportation costs of raw material, parts, tools, work-in-process, and finished products among the facilities, facilitating the traffic flow and minimizing the costs of it, maximization of the layout performance, minimization of the dimensional and form errors of products depending on the fixture layout, minimization of the total number

of loop traversals for a family of product, increasing the employee morale, and minimization of the risk of injury of personnel and damage to property, providing supervision and face-to-face communication. (Azadeh, et al.)

Research has shown that "striving for higher production efficiency have entailed the need for applying more efficient modular-type systems of specified material flow patterns" (Siemiatkowski, et al., 2019). This has led to an increase in the use of "flexible manufacturing systems (FMSs) [which] are required to adapt to changes in product mix, demand, and designs with low-cost solutions" (Zhao and Wallace, 2014).

2.2 - Workflow Congestion

In a manufacturing setting "workflow congestion is a major concern in a facility because it results in immediate safety problems: damaged uprights, damaged vehicles, and damaged workers" (Zhang, et al., 2009). Machine locations in a plant can be crucial in cutting time and therefore costs. As Nyemba and Mbohwa (2017) explained, "A complex or multi-stage manufacturing system must follow a clearly defined path with very little or no backtracking to produce high quality products within the shortest possible time". Initially, in a Zimbabwaean furniture plant, bunk beds traveled back and forth across the factory numerous times before reaching complete form. In order to improve metrics like throughput time, productivity, and efficiency, machine distance matrices were created and updated as the machine layout was restructured (Nyemba and Mbohwa, 2017). After updating the machine distance matrix to show the latest improvements, “The comparisons were carried out for the components of the company’s main products and showed an average of 43% reduction in transportation distances, an improvement in the overall production process” (Nyemba and Mbohwa, 2017). We believe using a machine distance matrix would be very beneficial for our client, due to the fact that the product is going back and forth between two buildings twice in the manufacturing process.

2.3 - Non-Value Added Functions

While observing the process for creating the bench, specifically the steps for gluing and adding the upholstery, the team noticed a few non-value added movements. This problem has been noticed in other furniture warehouses in the past, which “Direct labor cost was becoming a major part of the company’s cost structure. Excess handling… [was] driving inefficiencies up” (Klemperer, et al., 2003). Sanchez, et al. (2020) improved a manufacturing process for stainless steel pipes by identifying and removing non-essential steps. For example, Sanchez, et al. (2020) identified that “[The Non-Value Added steps] could be eliminated using dedicated carts that are equipped to store the quantity of material being transported in one trip.” Similarly, we believe that purchasing an additional cart, bringing the total to four, would help improve efficiencies since benches are stacked in columns of four.

2.4 - 5S Model

For improving workplace organization, we are interested in implementing the 5S model. 5S stands for “Sort”, “Set in Order”, “Shine”, “Standardize”, and “Sustain” (Sanchez, et al., 2020). With the sort step, we feel we can reorganize the large columns of incomplete bench bases to create more space for gluing and upholstering the bases. Sanchez, et al. (2020) planned on a similar strategy stating “each area will be sorted, removing waste and unnecessary items from the area.” Set in order is another step which could yield great results for our client, possibly moving the gluing and upholstering steps to the same table to prevent unnecessary movement of the heavy base. Sanchez, et al. (2020), despite having a different product to manufacture, implemented a similar idea, stating “set in order will be accomplished by rearranging each area so that tools are arranged in an optimal pattern to avoid over-movement.”

2.5 - Finite Element Analysis

Overall, an area where the team feels we can make an immediate impact is in altering the materials used in the manufacturing of the bench. One method we found in our research for “faster, less costly, and more optimized product development” is the Finite Element Method (FEM), or Finite Element Analysis (FEA) (Tankut, et al., 2014). Finite Element Method tests a product without costly machinery or numerous sample products, “[allowing] the removal of the indefiniteness before the manufacturing and making the decisions related to manufacturing in a more healthy and economical manner” (Koç, et al., 2011). Finite Elements are considered “disjoint (non-overlapping) components of simple geometry” (Tankut, et al., 2014). Strength at these small components is aggregated to determine “the overall performance of the structure” (Chen and Wu, 2018). The FEM involves Industrial and Systems Engineering course concepts such as optimization and linear programming, using “integral formulations rather than difference equations to create a system of algebraic equations.” (Tankut, et al., 2014).

These calculations tend to be solved in some type of Computer-Aided Design (CAD) software, where “strength calculations of the designed product could be made by means of the computer aided structural analysis software” (Tankut, et al., 2014). As Wengang, et al. (2019) describes the rise of virtual product testing, “with the development of computer technology and finite element (FE) theory, it is more convenient now than in the past for researchers to analyze complex structures with FE software.” Specifically in the field of furniture, “in the case of carcass furniture, significant savings can be achieved by minimizing dimensions of the cross section of wood elements” (Tankut, et al., 2014). However, one has to be careful making the wood components too thin. As Lipinskis and Spulle (2011) warns, “the thinner the plywood is, the bigger differences in mechanical properties between plywood of the same thickness and of different face veneer grain direction appear. “

We believe we can use FEA to calculate the least amount of wood necessary capable of maintaining the specified load requirements or find an alternative type of wood to the Russian Birch currently used by the client, one that is both lighter and cheaper. As Güray et al (2015) describes improving furniture design, “Optimum design of the furniture can be achieved by i. changing the raw material, ii. changing the geometry of the sections of the members and iii. thickening the cross-sections and shortening the member sizes.” This type of outcome is realistic based on other uses of FEM in the field, with a study on “improvement in the existing designs of wooden desks and chairs [making] it possible to convert low grade logs, such as thinning materials, into environmentally and ecologically friendly school facilities” (Tankut, et al., 2014). Even wood composites have been found to have comparable strength properties at a lower cost. As Kasal (2006) determined, “the sofa frames constructed of beech and the sofa frames constructed of [Okoume plywood] have given close strength values. Therefore, [Okoume plywood] can be preferred to the solid beech and solid pine due to its many technical and economical advantages such as stability and feasibility.” A comparable strength value might be good enough considering the weight and rigidity of the client’s bench in its current state. Rundgren and Wörmke (2011) encountered a similar situation when helping IKEA improve a sofa frame with FEA: “because the current frame is such a rigid design there is a very small risk for failure. This is possibly due to over dimensioning.”

Even with all of the reasons to use Finite Element Analysis, there are some shortcomings to the process. Unless one has the tools to conduct strength testing on their own materials, they must make assumptions regarding their mechanical properties. Rundgren and Wörmke (2011) ran into this type of issue with the following: “Having the members in the design modelled as sheet metal would allow for automatic interpretation of shell surfaces which could greatly improve analysis speed. Not satisfied with the limitations the shell interpretation came with the parts were re-interpreted as solids manually. This produced incorrect analysis results.” Thus, they encourage the use of FEA with the following guideline: “The intention of performing FEA is not to eliminate the real tests but rather to reduce the time for a product to pass through the process” (Rundgren and Wörmke, 2011).

In order to determine an appropriate threshold for Factor of Safety, we turned to existing literature in the furniture field. According to Chowdary et al. (2019), “A factor of safety (FOS) of 1.0 indicates that the material has just begun to exhibit plastic deformation. As a result, the minimum factor of safety should have a value that is greater than 1.0. Higher FOS is always favourable.” For reference, the definition of plastic deformation is “the permanent distortion that occurs when a material is subjected to tensile, compressive, bending, or torsion stresses that exceed its yield strength and cause it to elongate, compress, buckle, bend, or twist.” (Pfeifer, 2009). Since the damage under plastic deformation is permanent, we believe a cutoff for Factor of Safety of 1.0 would be appropriate for the bench we are working with.

Once we decided on the Autodesk Inventor software to conduct our Finite Element Analysis, we needed to determine their operational definition of factor of safety. On the Autodesk Inventor help page (2014), safety factor is defined as “the ratio of the maximum allowable stress to the equivalent stress (von-Mises), when using Yield Strength.” The Autodesk

Inventor help page (2014) also establishes the cutoff for an appropriate safety factor at 1.0, explaining "[Safety Factor] must be over 1 for the design to be acceptable. (Less than 1 means there is some permanent deformation)."

Chapter 3: Project Management

3.1 - Problem Solving Approach

For our project, we used the Finite Element Analysis to first determine if we can reduce the weight of the client's bench by testing the model without making any changes to their existing structural materials (in this case Russian Birch). Then, we assessed whether we can further reduce the weight of the bench by testing the model with other material properties, keeping in mind their overall cost and internal manufacturing processes. This included several different alternatives of testing. For instance, in one scenario, the team reduced all ¾ inch wood down to ½ inch wood, to see the structural impact this change would have. Another example would be hollowing out certain areas of the internal frame, or substituting the heavy steel feet for something lighter. In all of these scenarios, the team compared the results of the FEA to optimize which changes should be made without compromising the structural integrity of the model. These findings can then be used to determine which options best fit with the company's requirements before implementing the changes in production to reduce the weight of the benches.

3.2 - Requirements

CBBE defined their requirements with a prioritized list of wants, needs, and problems associated with this project in our first meeting with the company. The number one item on their list is that the benches are heavy and awkward to turn, flip over, adhere foam, assemble, upholster and manipulate throughout the production process, which is a potential safety hazard for workers. Thus, reducing the weight of the bench would serve to solve multiple issues at once. Next, the quality and aesthetics of the product must remain intact. CBBE believes that the visually appealing nature of the bench is a big part of its competitive edge and helped increase sales, so the general shape and style must remain intact. Also, a lot of their production processes are automated using heavy machinery such as the CNC router machine, and they would prefer to not have to make major changes to these programs.

As far as engineering requirements and specifications, CBBE is not required to meet certain dynamic or static load thresholds. Rather, the product is designed to sustain the weight of one or two individuals, not exceeding a total weight of 500 pounds. As such, all our testing was done using this maximum weight of 500 pounds to compare the stress induced on each model. Finally, any proposed changes must fall in line with the current materials order budget, as CBBE has negotiated a special price from their current supplier based on order size. If major material changes are to be implemented, they should be less expensive than the current economic ordering system.

3.3 - Gantt Chart

The Gantt Chart for this project contains all of the tasks completed by the group during each phase of the project. This chart highlights how many days were spent on each of the tasks. For simplicity, these large images are located at the end of the report. Please see Appendix F: Gantt Chart for an up to date schedule.

3.4 - Flow Charts

The System Block Diagram can be found in detail in Appendix D, but can be summarized as follows: the initial cutting of the wood using the CNC router, screwing together of the base, attachment of the foam using glue, wrapping of the upholstery to the base and back, attachment of the back and feet to the base, crating of two stacked benches, and shipment. Like a process flow map, this diagram allows the team to break down each and every step of the process in order to isolate critical areas and ensure that every part of the process is accounted for. Please see the figure for more details.

3.5 - Budget

The client has not allocated any money to the KCC team, however, we can still perform a cost-benefit analysis on the different possible solutions. The engineering budget can be found in Appendix E and is subject to change as additional work is needed. The budget does not include time spent writing the report itself. The project is for the mutualistic benefit of the students and the company, with no currency or other means being exchanged besides knowledge. However, we have gathered estimates for certain equipment that could improve the production process.

We believe an overhead crane would make it a lot easier to move incomplete and complete benches across the building where most of the production is done. An overhead crane would cost approximately $50,000 based on competitive internet pricing. Additionally, KCC believes a logical purchase would be the addition of a fourth moving cart to the gluing stations. Currently, the company owns three moving carts but stacks the incomplete benches in columns of four. The average price for a moving cart capable of supporting the weight of a bench would be around $200. The budget also accounts for the price of the Autodesk Inventor software being used by the team for simulation and design purposes. Students have access to most of these resources for free, but in an industry project these softwares would need to be purchased.

3.6 - Team Assignment and Overall Schedule.

Team assignment is as follows: Garrett Smith is the Project Manager and Process Engineer. Ernie Rivera is the Project Coordinator and Systems Engineer. Jake Santa Cruz is the Data Analyst and Quality Engineer. Austin Hester is the Technical Expert and Operations Research Engineer. The Overall schedule is presented in the form of a Gantt Chart which can be found in Appendix F. This chart is subject to change throughout the course of the project as new work is added.

3.7 - Available and Required Resources

The VP of Special Projects has granted the team permission to conduct onsite visits as needed (with a one-day prior notice) and is available by phone or email to assist the KCC team. The Director of Engineering gave the team access to the designs/ blueprints of the product as well as access to the CNC machine programming software. The software that the group utilized was Autodesk Inventor to conduct the Finite Element Analysis.

Chapter 4: Proposed Solutions

4.1 - Crane

The group came up with a few potential solutions to the variety of areas of improvement our client possesses. An overhead crane is one way of approaching the weight issue of the benches. Rather than moving the benches by hand and risking injury, employees could use the crane to bear the vast majority of the weight. However, this method is very costly and would only ease the load of the benches some of the time due to the multi-building setup the client uses.

4.2 - Workflow Optimization

Another approach considered by the group is workflow optimization. Improving the manufacturing system would result in lower cycle times, and therefore less labor costs. However, it is highly unlikely the exact same system will be used when the company moves buildings in Q1 of 2021, making it hard for our group to predict what steps will carry over in the move.

4.3 - Finite Element Analysis

Our last proposed solution is Finite Element Analysis, or FEA. If done properly, this method would result in weight, cost, and waste reduction, a major breakthrough in the manufacturing process. These results would also carry over as the company moves locations.

However, FEA is just an approximation, requires expensive software, and took a semester's worth of careful training to learn. Proposed solutions are simplified in the table below:

Table 1: Proposed Solutions		
Solution # / Name	**Benefits**	**Disbenefits**
1) Warehouse Crane	Less lifting and turning for warehouse employees means fewer injuries	Solution is very costly, the price of the crane alone is $50,000. CBBE has multiple facilities on site, the crane would only be useful in one
2) Workflow Optimization/ Warehouse Optimization	Reduction in cycle time = lower production costs.	CBBE is soon relocating, so these warehouse design changes and employee training could be lost in transition
3) Finite Element Analysis (FEA)	Weight, cost, and waste reduction	Requires expensive software, More accurate models take more time to compute, technical learning curve

Chapter 5: Challenges Faced

5.1 - Initial Design

At the initial phases of this project, the KCC team was considering multiple systems engineering approaches to help the company reduce costs. This includes workflow optimization, warehouse optimization, time studies, and other alternatives to help CBBE with their production. However, the main concern from the company was to reduce the weight of the benches, and they didn’t want to make major changes to their programs. CBBE’s number one item on the list of problems was that the benches are heavy and awkward to manipulate during the assembly process, which is a safety hazard for employees. Additionally, since they are relocating soon, we wanted to be sure that our implementations would have a lasting impact, even if the layout of the warehouse was completely different. Thus we decided to try to reduce the weight of the bench as much as possible through the use of Finite Element Analysis (FEA).

5.2 - Prior Experience and Software Difficulties

Being a team composed of industrial and systems engineers, we knew that this would be a challenge for us. The team was familiar with programs such as AutoCAD and SolidWorks, but no members of the team had taken a full course related to FEA, so this was relatively new territory. We knew the desired results could be achieved based on the extensive research we conducted, however we wanted to be sure that we would be able to implement it ourselves, even with our limited experience. To do so, we needed to teach ourselves a fair amount about FEA and how to use these programs. We started by obtaining the 3D model from the company and running some initial tests based on what we had learned from online sources and videos. The 3D model files obtained from CBBE did not contain screws. There was a separate file that contained all the screw holes for the machining phases of production. However, this was not compatible with Inventor. As such, the 90+ screws had to be tediously added by hand before any tests could be run.

5.3 - Virtual Model vs. Physical Model

In order to make sure these additions were accurate, we decided to compare the virtual model with the physical model. The team obtained a sample bench unit from CBBE to take home and study. It was at this point that the team noticed one glaring issue. The physical model that had been produced and assembled by CBBE contained many differences when compared to the virtual model that was being used for production. For example, the physical model contained 113 screws, whereas the virtual model only contained 92. There were other issues including the dimensions being slightly different than they were listed to be on the virtual model. Thus, the team decided to continue with the virtual model, but made sure to keep track of any discrepancies so they could be pointed out to the company.

5.4 - Preliminary Tests and Seeking out Experts

Finally, the team was able to run some preliminary tests on the 3D model to determine the base conditions and limitations of the bench model that was currently in production. We obtained good results, but they were a bit tricky to decipher since we didn't have much experience with this type of analysis. Also, we needed to make sure that we were doing it correctly and getting accurate results before proceeding and making inferences based off of this data. To do so, the team sought out the help of faculty members at KSU and employees at CBBE. The team enlisted the help of Professor Santana Roberts from the Mechanical Engineering department at KSU. We met with him virtually, and then in person, to validate our results and simplify the underlying concepts. Additionally, Dennis Batin is the Director of Engineering at CBBE and has made himself available to help the team should we run into any problems with the design or the model itself. By taking advantage of the great resources available to students at KSU, the team was able to overcome these challenges in order to continue successfully with the project.

Chapter 6: Methodology

6.1 - Meeting With Client

Our team decided to focus on optimizing the amount of wood needed in order to make the bench lighter. We decided to use the Finite Element Analysis (FEA) method to accomplish this goal. Our client had not yet provided us with enough information for the bench in order for our team to tackle this type of analysis, so we had to schedule another meeting with the client to discuss the path ahead and acquire more information.

6.2 - Obtained Data

The client confirmed for us that they had not yet performed this type of analysis on the bench and were curious to see what kind of results we could accomplish. They provided us with the info necessary for us to do our analysis. We received a copy of the latest version of the CAD model from their Director of Engineering, the routing instructions they used for the CNC machine, and the list of materials needed to ascertain their properties.

6.3 - Autodesk Inventor

After the acquisition of the data, our team then had to choose which program we were going to use in order to conduct our analysis. In our earlier report, we had mentioned several possibilities such as DesignSpace, FEMAP, Working Model FEA, COSMOS, and Solidworks, but we proceeded with the analysis using Autodesk Inventor because it is in the same modeling family that the client uses. This allowed the team to focus on learning the program's features and saved a lot of time in wasted efforts to convert the files into different programs.

6.4 - Developing Models

We began by isolating the 3D model in AutoCAD from the other portions of the workspace. After opening the model in Autodesk Inventor, it was necessary to label all 35 parts and to organize those labeled parts into sections of the bench that could be easily located and grouped. The part labels are as follows; Right Bracket, Left Bracket, Left Frame, Back Frame, Front Frame, Right Frame, Center Frame, Back Support (BS) 7, Top of Back, BS (9,3,5), Front of Back, Left of Back, Right of Back, BS (2,4,1,6,8), Center Back Support, Left Back Support, Right Back Support, Back of Back, Back of Base, Bottom of Back (later removed as a spacer), Sides of Base, Top of Base, Back Left Foot, Back Right Foot, Front Right Foot, Front Left Foot, Right Foot Support, and Left Foot Support. Each part was then assigned a material, either Wood (Birch), Medium Density Fiberboard, or Steel (Mild, Welded), according to the material used in the physical models.

Changes. After importing the model into Inventor, several small changes needed to be made to more accurately match the current model in production. These included hollowing out the foot supports, as the original file had solid pieces of steel, whereas in reality, they are hollow with 16 gauge (0.0625") steel as the frame. Secondly, a ¾ inch section of steel needed to be added to both long ends of the foot supports to allow for holes to be drilled and bolts to affix the piece to the baseplate.

Holes. The next step in developing the models was to add holes for the screws used to hold the pieces together. This process involved utilizing a second AutoCAD file that contained pilot hole locations drilled by the CNC machine. By calculating the various distances from the edge of the pieces to the array of holes, and again between the holes, we were able to create a 2D sketch on the various parts in Inventor. The Hole tool was then applied to the specified locations, creating a countersunk hole with a head diameter of .332 in, an angle of 82 degrees, a total length of 1.5 inches, and a shaft diameter of .164 in. These dimensions match the dimensions of a Number 8 Wood Screw, the same used in the assembly of the benches. As mentioned above, a few of the dimensions were not completely accurate when transferring from the AutoCAD model with the pilot holes to the Inventor Model, and new calculations had to be made to ensure that the screws were placed in the center of the support beams. Using the same method, holes were created in the foot supports, however these holes were standard holes with a diameter of .265 in, and a length of 1in. These correspond with the ¼”-20 Hex Bolts used to affix the foot supports to the baseboard, as well as the back braces to the backboard.

Screws/Bolts. After the holes were created in the model, a screw/bolt needed to be constrained into each one. An Inventor part file was downloaded matching the specifications needed for each fixture. A Number 8 Wood Screw file (Petitt, 2006) was downloaded from an

Autodesk Inventor forum, a ¼” -20 Hex Bolt file (Lunchbox, “CAT Hex Head Bolt- metric”, 2012) was downloaded from GrabCAD Community, ¼” Flat Washer file (Lunchbox, “CAT Flat Washer”, 2012) was downloaded from GrabCAD Community, and a ¼” T-Nut file (Mansfield, 2012) was downloaded from GrabCAD Community. After each of the above parts was loaded into the assembly, each was constrained into their appropriate holes. For the screws, each was firstly along the rotational axis to the corresponding axis in each hole. Each screw was then constrained forcing the head of the screw to match the same plane as the surface. Each T-Nut was firstly constrained along the rotational axis to the corresponding axis in each hole. They were then constrained so that the back of the nut was along the same plane as the surface. Each washer was constrained so that the length of the screw was within the hole for the washer, and so that the face of the washer was constrained to the bottom of the bolt head. Each bolt was constrained so that the rotational axis of the part lined up with the rotational axis of the T-Nut. Finally, each bolt head/washer face was constrained so that it was affixed to the surface of the appropriate part.

Glue Constraints. After all of the bolts and screws were properly inserted and constrained, the individual parts in the backrest of the bench needed to be constrained, mimicking the action of glueing the parts together. To do this, each back support was constrained to the adjoining part in all three dimensions so that they would stay together. Each support was also constrained to the back, front, top and all three bottom pieces. Again, these constraints act as the action of gluing these individual parts together.

6.5 - Finite Element Analysis

After developing the model according to the specifications above, we began Stress Analysis on the assembly. We began by applying a fixed constraint to all four foot pieces. The Fixed constraint prevents any translation or rotation of the parts, and therefore prevents slipping and rotation in the simulation. After the assembly was constrained, a 500lb force was applied to the top of the seat, and the simulation was run. The simulation entails creating a mesh of the assembly and applying the above force repeatedly, and producing a series of results. These included the Mass, Von Mises Stress, Displacement, Safety Factor, and several other stresses and strains that were not required for the analysis. After running this simulation, we ran two alternative simulations on all iterations, including one with two forces of 250lb each applied 12.7519" from the center long way on the bench to simulate two people sitting. The value of 12.7519" was found from Iterations 5-7 to line the weight up on the support. The third simulation that was run involved using the split face tool, to create two 12"x12" squares on the faces of the benches and then apply an even pressure along that foot. This is the most accurate simulation of someone sitting on a bench because it creates a smoother force application over a larger area. The pressure used was 1.736 psi which was calculated by dividing 250lb by the 144 inch area.

These values can be found in the next chapter and served as our baseline analysis, due to the company not providing any baselines.

6.6 - Iterations

Iteration 1 (Original/Solid Feet). Before producing our baseline results, we ran a simulation where the bolts were not included in the model. The model contained only the screws holding the birch pieces together and can be found in **Figure 4**. This simulation was run before we realized that we needed to add the bolts and was kept for comparison's sake.

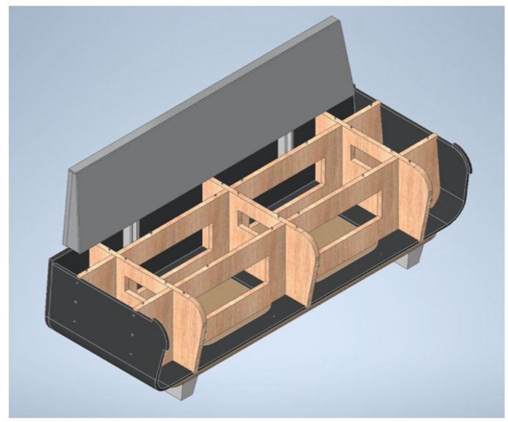


Figure 4: Iteration 1 - Original Solid Feet

The same constraints and forces were applied, and the simulations were run. As stated before, these results can be found in the next chapter.

Iteration 2 (Bolted Baseline w/ Hollow Feet). Our second alteration consisted of adding bolts to the foot supports and the back brackets. The first step was to hollow out the foot supports because they were solid steel in the virtual model, but in reality they were made of 16 gauge steel which is 1/16 of an inch thick (0.0625”). The supports were made hollow using the shell function, which hollowed all sides to a set thickness of 0.0625” and we proceeded to add a lip to both long edges to drill holes and insert bolts. To create this lip, we created a midplane between the two short edges, and we drew a sketch from that top edge .75” out from the top, and down

0.0625” - the thickness of the steel - and back to the support. We then used the extrude tool to expand that sketch to one end of the support. We then used the mirror tool to mirror that extrusion across the sketch plane, which created a lip on one side of the support. We created another midplane between the two long edges of the foot support and then again used the mirror tool to copy the lip to the other side of the support. We used the hole tool to create holes along the bottom edge of the lips to both foot supports. Those holes were simple holes 0.265” in diameter and 1” deep. A model was not shown here because the interior components match that of **Figure 4**. The changes made to the foot supports can be found in a comparison in **Figure 5**. After the holes were drilled, we inserted and constrained the T-nut to the baseboard and the outside of both back pieces, the back of the back and the back of the base. We constrained each T-nut within the hole, and then to the surface. For the foot supports, we then constrained a bolt into each T-nut and then constrained the head of the nut to the bottom of the foot support. For the back supports, we included a washer because the head of the bolt was too small to reach the sides of the back brackets. Each bolt was constrained into a washer and the washer was constrained to the bottom of the bolt head. Each

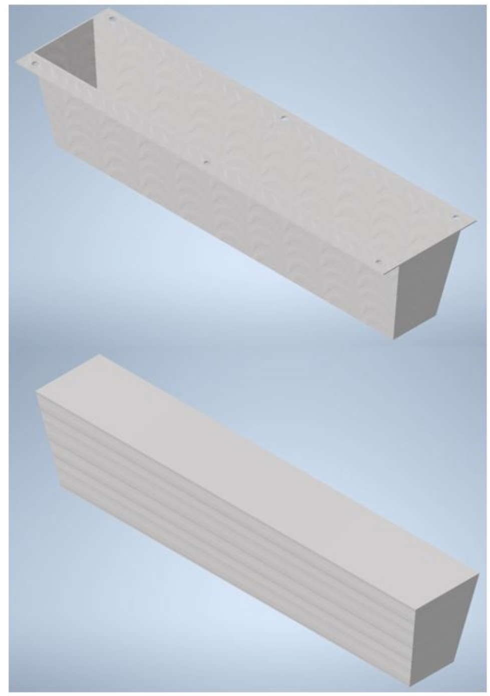


Figure 5: Iteration 2 – Bolted Baseline with Hollow Feet

bolt was then constrained into the T-nut and the bottom of the washer was constrained to the back brackets. Simulations were run and the results can be found in the next chapter.

Iteration 3 (Plywood). After producing our baseline results, we were free to make changes to the model, re-run the simulation, and compare results. Our first alteration involved changing the used wood to Plywood (Finish) as opposed to the original Wood (Birch). **Figure 6** shows this model, which looks very similar to the previous ones, however the wood has a slightly different color. The same constraints and forces were applied, and the simulations were run. As stated before, these results can be found in the next chapter.

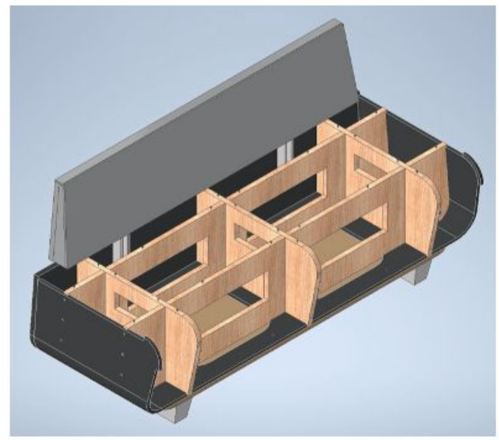


Figure 6: Iteration 3 - Plywood

Iteration 4 (½" Birch). Our fourth alteration consisted of returning to the original wood (Birch), however changing the thickness of all of the parts to ½" as opposed to the mix of ¾" and ½" present in the original model. This was accomplished using the Thicken/Offset command. For all interior parts, a thickness of 0.125" was taken from each face, to remove ¼", but retain the center of the part. For all exterior parts, a thickness of 0.25" was taken

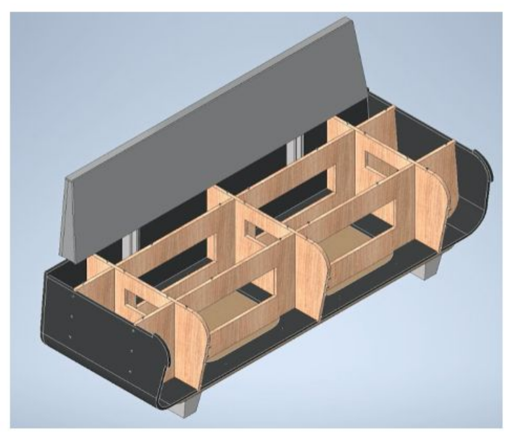


Figure 7: Iteration 4 – ½" Birch

from the outward face to retain the location of the interior face, in relation to the assembly. The following parts were adjusted; Front Frame, Back Frame, Left Frame, Center Frame, Right Frame, Base, Back Support (2,4,5,6,9), Back of Back, Left of Back, Right of Back, Left Back Support, Center Back Support, and Right Back Support. The following pieces were removed; Back Support (1,3,7,8). This interior of this model can be seen in **Figure 7**. We applied the same screws and bolts and then the same constraints and forces were applied, and the simulation was run. As stated before, these results can be found in the next chapter.

Iteration 5 (2x2 Support Beams). In this iteration, we built off of the previous ½” model and removed the center front to back support (center frame) and moved the remaining two (left frame and right frame) closer together. To accomplish this, the center support was deleted. We created a left to right midplane on the front support and then created a sketch on the front frame and drew the new cutout location for the front to back frames, and also created new holes to remove some weight. We used the Thicken/Offset tool to fill in all existing holes, and make the front frame solid. We then used the Extrude function to cut away the areas for the new holes and the frame, and finally used the mirror function to copy the extrusion to the other side. This same process was used on the back frame. We selected the front and back frames and made

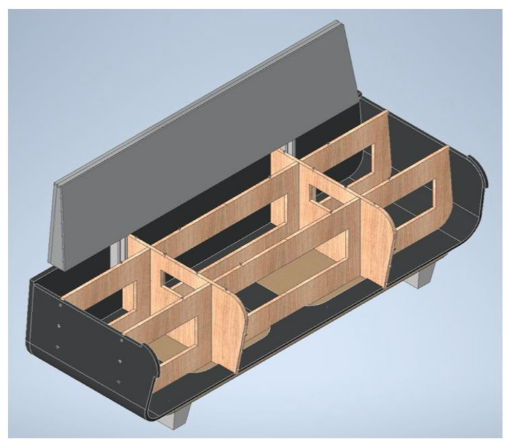


Figure 8: Iteration 5 – 2x2 Support Beams

them Grounded, which means that they wouldn't move for the next step. We constrained the front to back supports to the side to side ones in the new locations and then removed the grounded feature on the edited frames. This interior of this model can be seen in **Figure 8**. We applied screws to the new locations of the beams, and applied the same bolts as the previous iterations. We applied fixed constraints to the feet and applied all three force applications and ran simulations, results are found in the next chapter.

Iteration 6 (2x2 - Support Beams w/ Larger Holes). This iteration is very similar to the last one, where there are two front to back supports and two side to side supports in the base. The only difference is that the holes created for weight removal on the frames were made larger in this iteration using the Thicken/Offset tool. The left to right frames (front frame and back frame) were each edited by removing 1” from each of the four sides of the holes, therefore making the hole bigger by 2” in the vertical and 2” in the horizontal direction. This was applied to all three holes on the front frame and back frame. The left and right frames were edited by removing 5” from the bottom of the hole, since the original holes were small and near the top of the frame. The model was copied from the previous iteration, so screws and bolts were already in

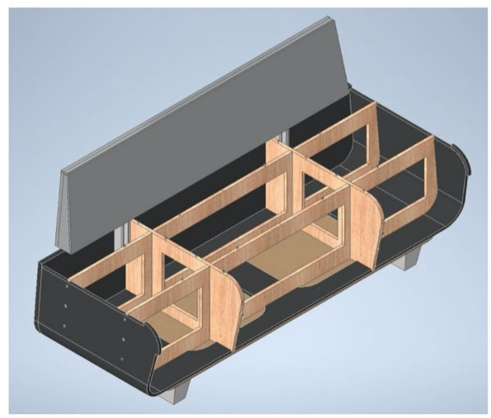


Figure 9: Iteration 6 – 2x2 Support Beams w/ Larger Holes

place. This interior of this model can be seen in **Figure 9**. The same constraints and force vectors were added and the simulations were run. The results are in the next chapter.

Iteration 7 (2x1). In this iteration which can be found in **Figure 10**, we built off of Iteration 5, however removed one of the left to right frames (Back Frame). The same general technique was applied. The Back frame was deleted, a sketch was created on the left frame. The new cutout for the side to side frame was centered along the bottom of the support and holes were drawn to remove weight. The original frame was filled in using the Thicken/Offset tool and then the sketch was Extruded into the left frame. The same process was applied to the right frame. This time, the Left and Right Frames were grounded and the Front frame was constrained into the new gaps. After this the frames were un-grounded and screws were added into the frames that were there. The same bolts were added and then the same Stress Analysis was run - feet were constrained and all three forces were applied. Results are in the next chapter.

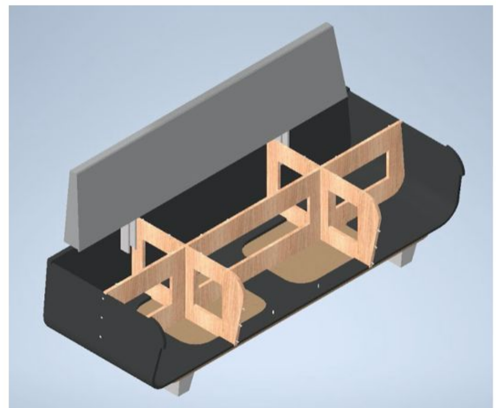


Figure 10: Iteration 7 – 2x1 Support Beams

Iteration 8 (1x1). In this iteration, we removed one more of the front to back supports so that there was only one front to back and one left to right. This iteration was built off of Iteration 7 and one of the front to back supports (Right frame) was removed. This time the remaining left to right frame (Front Frame) was edited using the previously described technique. This time the gap for the remaining support was made in the center and large holes were created on either side. This frame was grounded and the remaining front to back frame (Left Frame) was constrained into place and the frame was un-grounded. The screws were applied into the frames and the bolts were applied to their locations. This interior of this model can be seen in **Figure 11**. The feet were constrained and the forces were applied and all three simulations were run. The results are in the next chapter.

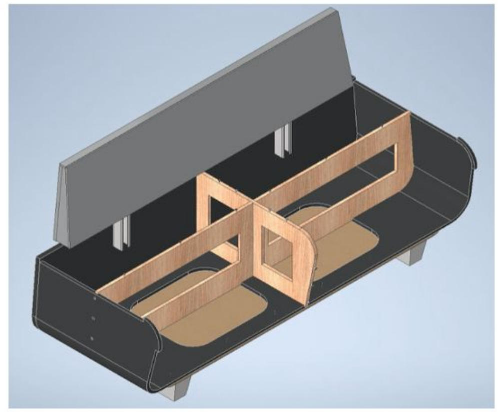


Figure 11: Iteration 8 – 1x1 Support Beams

Iteration 9 (Flat Beam). In this iteration, we decided to change the model completely and created a bench with two front to back beams that were only on the front and back edges and hollow from top to bottom in the middle. A horizontal ring that bisected the bench was used in place of the side to side supports. This model can be seen in **Figure 12**. The starting point for this model was Iteration 7. The front to back supports were edited in the same manner, and a sketch was created. We calculated the vertical midpoint of the support and moved 0.25” upward and downward to provide a gap for the horizontal ring. We calculated 5” inward from the front

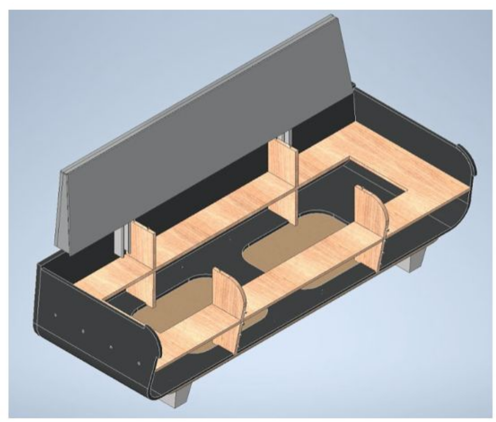


Figure 12: Iteration 9 – Flat Beam

and the back along the bottom of the gap and cut out the section in between (top to bottom). We left a cut of 0.75” on either side to lock into the horizontal ring. To create the ring, we edited the remaining side to side support (Front Frame) and closed all of the gaps using the Thicken/Offset tool. We used the same tool to add 30” to the top and sides to create a large workable rectangle. Due to the shape and rounded edges, this did not provide a large enough left to right workspace, so we created an extrusion and mirrored the shape, providing us with a large enough workspace. Since the sides are rounded and the front is rounded, we couldn’t leave the wood ½” like normal, we had to angle the sides of the board to match that of the exterior of the bench. To create this cut, we drew a sketch on the top of this frame, and another on the bottom of the frame, with the appropriate dimensions. We used the Loft tool, which creates a shape between two sketches. This sketch had matching cutouts of 0.75” to lock into the front to back frames. A large center hole was cut out to reduce weight and 5” were kept on all edges to match the other frames. We added screws to the frames and the bolts were added. We applied the constraints and the forces and ran the simulations, the results are in the next chapter.

Iteration 10 (Extra Support). To validate our results, we deemed it necessary to develop a model that included extra supports, which can be found in **Figure 13**. We could then trust our findings from removing supports if adding them produced opposite trends. To develop this model, we started from Iteration 2, our baseline, and added another support in both directions. To do this, we placed the Center Frame in the model a second time, and placed the

Back Frame in the model a second time. We applied the same process of edits to the Front Frame, to accommodate the extra frame. We created and edited a sketch, with four instead of three gaps for the supports. We also needed to edit the Front to back frames in the same way to accommodate an extra frame. The original side to side supports were grounded and the front to back supports were all constrained into place. Then the front to back supports were grounded and the remaining middle side to side frame was constrained into place. All frames were un-grounded and screws were placed into the supports. The bolts were added in their respective places. The feet were constrained, the forces were applied, and the simulations were run. The results can be found in the next chapter.

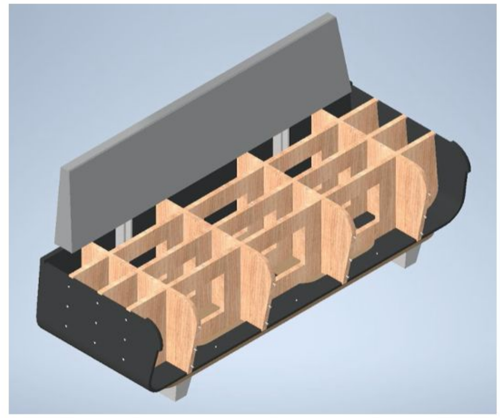


Figure 13: Iteration 10 – Extra Support Beams

Chapter 7: Results and Discussions

7.1 - Iteration Table Results

This chapter provides all of the pertinent results from the FEA simulations on the different iterations. Von Mises Stress in this case is used to determine if a material will yield or fracture. It is presented in ksi (kilo-pounds per square inch). Displacement is used to determine the distance an element moved from an original position to an ending position. Factor of Safety in this case is used to determine the measure of absolute strength in a structure relative to the applied load. It tells us the reliability of the design. **Table 2** provides the Mass, Von Mises Stress, Displacement, and Factor of Safety results from iteration 1, the Original (Solid Feet) model. The One Force column provides data from when there was one 500lb force applied to the center of the seat. The Two Forces column provides data from when there were two 250lb forces applied 12.7519" from the center. The Two Pressures column provides the data from when an even pressure was applied over the two 144 sq-in sections. A comparison of the application of the forces can be found in **Figure 14**.

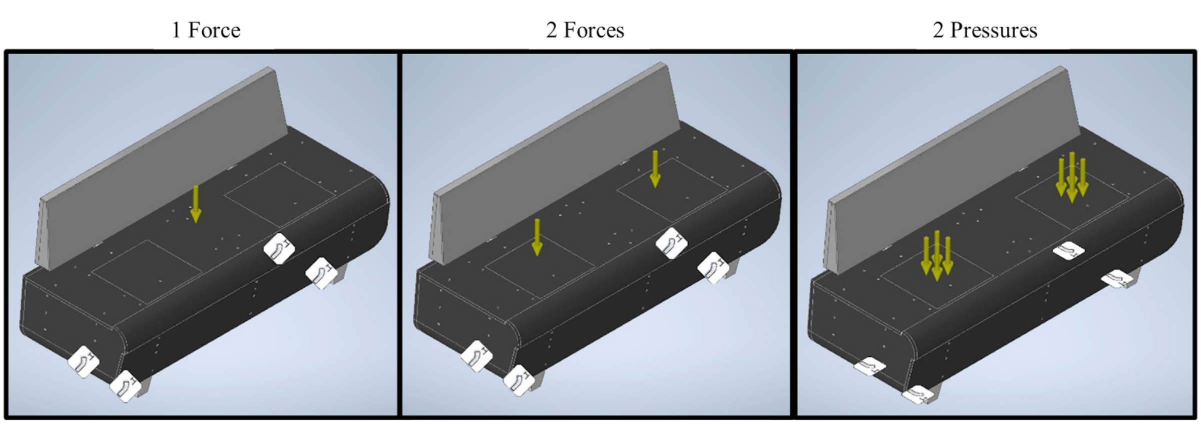


Figure 14: Comparison of Forces

Table 2: FEA Results for Iteration 1			
Original (Solid Feet)	1 Force	2 Forces	2 Pressures
Mass (lbmass)	197.738		
Von Mises Stress (ksi)	0.00000121056	0.000000939816	0.00000104019
	36.6546	36.9386	52.0827
Displacement (in)	0.020981	0.0230154	0.0352532
Safety Factor	1.38491	1.37426	0.974665
	15	15	15

The minimum and maximum values for Von Mises Stress increased as the different simulations were run. The displacement also increased as the different forces were applied. Conversely, the minimum safety factor decreased as the forces are applied differently, and the maximum safety factor remains the same. **Table 3** provides the same information but for iteration 2, the Bolted (Baseline) model.

Table 3: FEA Results for Iteration 2			
Bolted (Baseline)	1 Force	2 Forces	2 Pressures
Mass (lbmass)	113.806		
Von Mises Stress (ksi)	0.00000121056	0.00000154036	0.00000183028
	39.9205	41.4082	54.0499
Displacement (in)	0.0216597	0.0221554	0.0360032
Safety Factor	1.27161	1.22592	0.939191
	15	15	15

For the Bolted (Baseline) model, the minimum and maximum values for Von Mises Stress increased as the different simulations were run. The displacement also increased as the different forces were applied. Conversely, the minimum safety factor decreased as the forces are applied differently, and the maximum safety factor remains the same. **Table 4** provides the same information but for iteration 3, the Plywood model.

Table 4: FEA Results for Iteration 3			
Plywood	1 Force	2 Forces	2 Pressures
Mass (lbmass)	114.001		
Von Mises Stress (ksi)	0.0000019515	0.00000190355	0.00000212766
	46.2925	47.946	57.4712
Displacement (in)	0.0268606	0.0274745	0.0449122
Safety Factor	1.09658	1.05876	0.88328
	15	15	15

For the Plywood model, the minimum and maximum values for Von Mises Stress increased as the different simulations were run. The displacement also increased as the different forces were applied. Conversely, the minimum safety factor decreased as the forces are applied differently, and the maximum safety factor remained the same. **Table 5** provides the same information but for iteration 4, the ½” model.

Table 5: FEA Results for Iteration 4			
1/2"	1 Force	2 Forces	2 Pressures
Mass (lbmass)	90.4319		
Von Mises Stress (ksi)	0.00000240306	0.00000209014	0.00000257088
	175.359	172.215	231.023
Displacement (in)	0.0253611	0.0257971	0.0392747
Safety Factor	0.289482	0.294766	0.219733
	15	15	15

For the ½" model, the minimum and maximum values for Von Mises Stress increased from the application of One Force to the application of Two Forces, but decreased from the application of Two Forces to the application to Two Pressures. As in the previous tables, the value of the displacement increased as the different forces are applied. In this model, the minimum safety factor increased from the application of One Force to the application of Two Forces, but decreased greatly from the application of Two Forces to the application to Two Pressures. As in all of the previous models, the maximum safety factor remained the same. **Table 6** provides the same information but for iteration 5, the 2x2 model.

Table 6: FEA Results for Iteration 5			
2x2	1 Force	2 Forces	2 Pressures
Mass (lbmass)	90.5412		
Von Mises Stress (ksi)	0.000000203243	0.000000167889	0.0000000917936
	120.122	122.586	158.609
Displacement (in)	0.0246503	0.0252388	0.0380366
Safety Factor	0.422597	0.414103	0.320052
	15	15	15

For the 2x2 model, the minimum values for Von Mises Stress decreased as the different simulations were run. The maximum values for the Von Mises Stress increased as the different simulations were run. The displacement increased as the different forces were applied. Conversely, the minimum safety factor decreased as the forces are applied differently, and the maximum safety factor remained the same. **Table 7** provides the same information but for iteration 6, the 2x2 - Larger Holes model.

Table 7: FEA Results for Iteration 6

2x2 -Larger Holes	1 Force	2 Forces	2 Pressures
Mass (lbmass)	87.4919		
Von Mises Stress (ksi)	0.000000213577	0.000000165031	0.0000000891824
	121.119	127.035	170.711
Displacement (in)	0.0247797	0.025376	0.038255
Safety Factor	0.41912	0.399601	0.297364
	15	15	15

For the 2x2 - Larger Holes, the minimum values for Von Mises Stress decreased as the different simulations were run. The maximum values for the Von Mises Stress increased as the different simulations were run. The displacement increased as the different forces were applied. Conversely, the minimum safety factor decreased as the forces are applied differently, and the maximum safety factor remained the same. **Table 8** provides the same information but for iteration 7, the 2x1 model.

Table 8: FEA Results for Iteration 7			
2x1	1 Force	2 Forces	2 Pressures
Mass (lbmass)	84.8526		
Von Mises Stress (ksi)	0	0	0.000000385653
	211.253	218.848	323.501
Displacement (in)	0.0266597	0.0271411	0.039908
Safety Factor	0.240296	0.231957	0.156918
	15	15	15

For the 2x1 model, the minimum values for Von Mises Stress were 0 for the One Force and Two Force simulations, but increased slightly when the Two Pressures were applied. The maximum values for the Von Mises Stress increased as the different simulations were run. The displacement also increased as the different forces were applied. Conversely, the minimum safety factor decreased as the forces are applied differently, and the maximum safety factor remained the same. **Table 9** provides the same information but for iteration 8, the 1x1 model.

Table 9: FEA Results for Iteration 8			
1x1	1 Force	2 Forces	2 Pressures
Mass (lbmass)	83.0005		
Von Mises Stress (ksi)	0.00000234904	0.00000171787	0.00000215614
	140.06	140.332	137.993
Displacement (in)	0.0223209	0.0228042	0.0398104
Safety Factor	0.362439	0.361717	0.367867
	15	15	15

For the 1x1 model, the minimum values for Von Mises Stress decreased from the application of One Force to the application of Two Forces, but increased from the application of Two Forces to the application to Two Pressures. Conversely the maximum values for Von Mises Stress increased from the application of One Force to the application of Two Forces, but decreased from the application of Two Forces to the application to Two Pressures As in the previous tables, the value of the displacement increased as the different forces were applied. In this model, the minimum safety factor decreased from the application of One Force to the application of Two Forces, but increased from the application of Two Forces to the application to Two Pressures. As in all of the previous models, the maximum safety factor remained the same. **Table 10** provides the same information but for iteration 9, the Flat Beam model.

Table 10: FEA Results for Iteration 9			
Flat Beam	1 Force	2 Forces	2 Pressures
Mass (lbmass)	86.1447		
Von Mises Stress (ksi)	0.000000321216	0.000000321461	0.000000210534
	114.065	239.846	358.212
Displacement (in)	0.0284551	0.0291895	0.0423726
Safety Factor	0.22177	0.211649	0.141713
	15	15	15

For the Flat Beam model, the minimum values for Von Mises Stress decreased as the different simulations were run. The maximum values for the Von Mises Stress increased as the different simulations were run and the displacement values also increased as the different forces were applied. Conversely, the minimum safety factor decreased as the forces are applied differently, and the maximum safety factor remained the same. **Table 11** provides the same information but for iteration 10, the Extra Support model.

Table 11: FEA Results for Iteration 10			
Extra Support	1 Force	2 Forces	2 Pressures
Mass (lbmass)	124.761		
Von Mises Stress (ksi)	0.000000824151	0.000000918384	0.00000179695
	14.8684	15.8301	26.4273
Displacement (in)	0.0155733	0.01589	0.0327171
Safety Factor	3.41417	3.20676	1.92086
	15	15	15

For the Extra Support model, the minimum and maximum values for Von Mises Stress increased as the different forces were applied. As in the previous tables, the value of the displacement increased from One Force to Two Forces to Two Pressures. In this model, the minimum safety factor decreased as the different forces were applied. As in all of the previous models, the maximum safety factor remained the same.

7.2 - Comparative Table Results

After computing the results for each iteration, it became necessary to compare the results within each force application. **Table 12** below contains the summary of all 10 iterations while applying one force of 500lb in the center of the bench.

Table 12: FEA Results with 1 Force Applied							
Iteration	1 Force	Mass	Von Mises Stress (ksi)		Displacement	Safety Factor	
		(lbmass)	Min	Max	(in)	Min	Max
1	Original (Solid Feet)	197.738	0.00000121056	36.6546	0.020981	1.38491	15
2	Bolted (Baseline)	113.806	0.00000176702	39.9205	0.0216597	1.27161	15
3	Plywood	114.001	0.0000019515	46.2925	0.0268606	1.09658	15
4	1/2"	90.4319	0.00000240306	175.359	0.0253611	0.289482	15
5	2x2	90.5412	0.000000203243	120.122	0.0246503	0.422597	15
6	2x2 -Larger Holes	87.4919	0.000000213577	121.119	0.0247797	0.41912	15
7	2x1	84.8526	0	211.253	0.0266597	0.240296	15
8	1x1	83.0005	0.00000234904	140.06	0.0223209	0.362439	15
9	Flat Beam	86.1447	0.000000321216	114.065	0.0284551	0.22177	15
10	Extra Support	124.761	0.000000824151	14.8684	0.0155733	3.41417	15

As we can see, the mass decreases from the iteration with solid feet to all subsequent iterations. As more wood and supporting beams were removed, we can see a trend of decreased mass from iteration to iteration. It is important to note that in iteration 10, with the introduction of extra supports, we see an increase in the mass of the bench, providing validation in our study of the mass reduction. In addition, the smallest mass was achieved on iteration 8, the model containing one support beam in either direction. **Figure 15** shows the minimum values for the Von Mises Stress across the 10 iterations. The higher values correspond to the bench models 1,2,3,4, and 8, whereas the lower values correspond to the bench models 5,6,7,9, and 10. The values for the maximum Von Mises Stress can be found in **Figure 16**, but do not exhibit the same pattern as the minimum values. The values for the displacement are generally consistent with the exception of the minimum displacement on iteration 10, which had extra supports and the maximum displacement on iteration 9, where the horizontal beam was modeled. **Figure 17** provides a graph of the minimum Safety Factors, which can be seen exhibiting an overall decreasing trend, with the exception of our

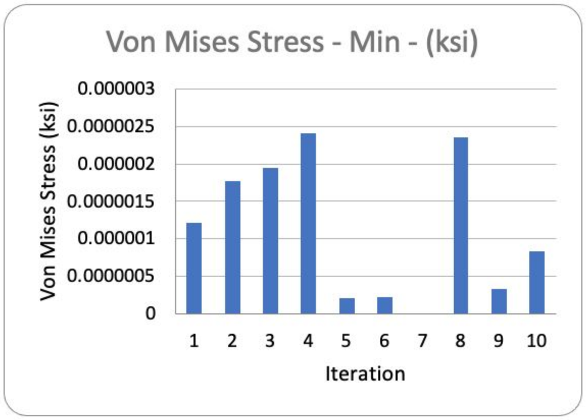


Figure 15: One Force Von Mises Min

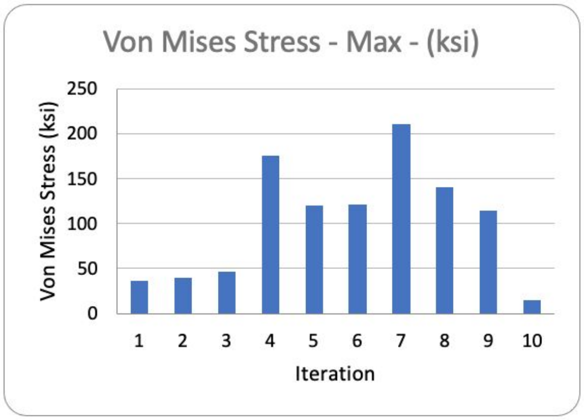


Figure 16: One Force Von Mises Max

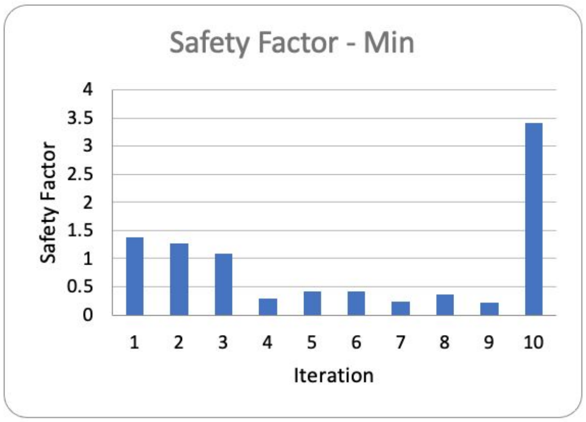


Figure 17: One Force Safety Factor Min

validation study (iteration 10), which provided a large increase. **Table 13** below provides the same information provided in **Table 12**, but for the simulation where two 250 lb forces were applied to the top of the seat at the location of the cross-beams from iterations 5,6,7.

Table 13: FEA Results with 2 Forces Applied							
Iteration	2 Forces	Mass	Von Mises Stress (ksi)		Displacement	Safety Factor	
		(lbmass)	Min	Max	(in)	Min	Max
1	Original (Solid Feet)	197.738	0.000000939816	36.9386	0.0231054	1.37426	15
2	Bolted (Baseline)	113.806	0.00000154036	41.4082	0.0221554	1.22592	15
3	Plywood	114.001	0.00000190355	47.946	0.0274745	1.05876	15
4	1/2"	90.4319	0.00000209014	172.215	0.0257971	0.294766	15
5	2x2	90.5412	0.000000167889	122.586	0.0252388	0.414103	15
6	2x2 -Larger Holes	87.4919	0.000000165031	127.035	0.025376	0.399601	15
7	2x1	84.8526	0	218.848	0.0271411	0.231957	15
8	1x1	83.0005	0.00000171787	140.332	0.0228042	0.361717	15
9	Flat Beam	86.1447	0.000000321461	239.846	0.0291895	0.211649	15
10	Extra Support	124.761	0.000000918384	15.8301	0.01589	3.20676	15

In the case of two forces, since the models were the same, with only the application of the force changing, the masses remained the same. With the application of Two Forces, the minimum values for the Von Mises followed the same

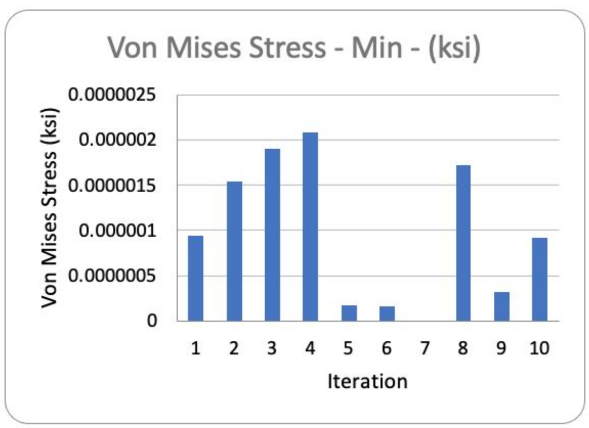


Figure 18: Two Force Von Mises Min

general pattern of values, with higher values on models 1,2,3,4, and 8, and the lower values corresponding to models 5,6,7,9, and 10. This graph of values can be found in **Figure 18**. Again, the same general pattern exists for the maximum values of Von Mises Stress which can be found in **Figure 19**, and in this second simulation, however there is a new maximum value in iteration 9. As before, the minimum value of the safety factor produces an overall downward trend that can be seen in **Figure 20**, with the exception of our validation study, iteration 10, which produces an overarching maximum value. **Table 14** below provides the same information provided in **Table 13**, but instead of applying the 250 lb forces in two specific points, the weight was spread evenly across two areas of 144 square inches each. This would help spread the weight evenly across the area to best simulate people actually sitting on the bench.

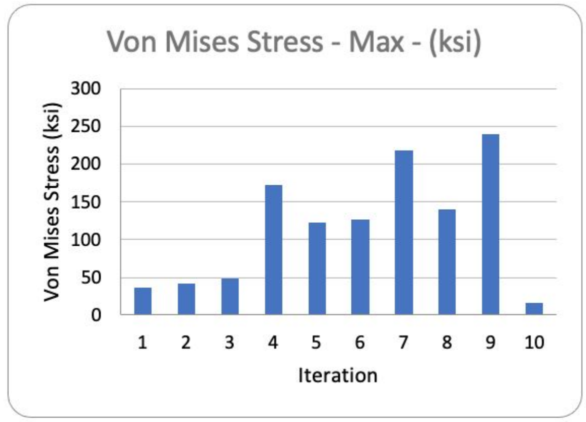


Figure 19: Two Force Von Mises Max

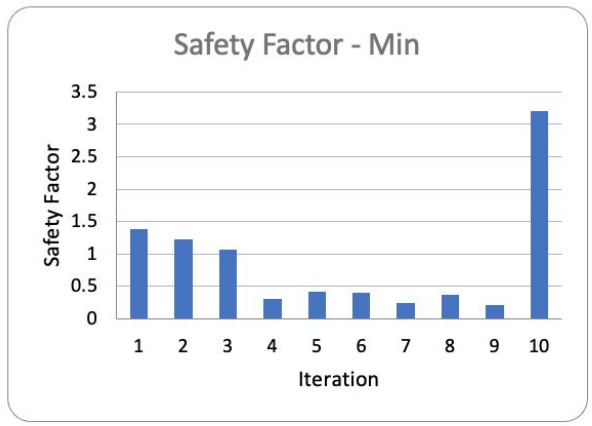


Figure 20: Two Force Safety Factor Min

Table 14: FEA Results with 2 Pressures Applied							
Iteration	2 Pressures	Mass	Von Mises Stress (ksi)		Displacement	Safety Factor	
		(lbmass)	Min	Max	(in)	Min	Max
1	Original (Solid Feet)	197.738	0.00000104019	52.0827	0.0352532	0.974665	15
2	Bolted (Baseline)	113.806	0.00000183028	54.0499	0.0360032	0.939191	15
3	Plywood	114.001	0.00000212766	57.4712	0.0449122	0.88328	15
4	1/2"	90.4319	0.00000257088	231.023	0.0392747	0.219733	15
5	2x2	90.5412	0.0000000917936	158.609	0.0380366	0.320052	15
6	2x2 -Larger Holes	87.4919	0.0000000891824	170.711	0.038255	0.297364	15
7	2x1	84.8526	0.000000385653	323.501	0.039908	0.156918	15
8	1x1	83.0005	0.00000215614	137.993	0.0398104	0.367867	15
9	Flat Beam	86.1447	0.000000210534	358.212	0.0423726	0.141713	15
10	Extra Support	124.761	0.00000179695	26.4273	0.0327171	1.92086	15

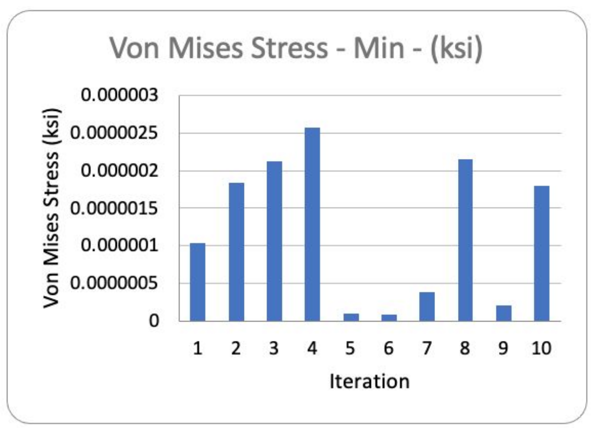


Figure 21: Two Pressures Von Mises Min

As with the previous set of results, the same models were used, so the mass values are the same as before. In this application, the minimum values for the Von Mises Stress followed the same pattern as before, further validating our results, however the value for iteration 10, has increased much more as compared to its previous force applications. This can be seen in **Figure 21** where the higher values correspond to iterations 1,2,3,4,8, and 10, and the lower values correspond to iterations 5,6,7, and 9. The values for the maximum Von Mises Stress can be found in **Figure 22**, following the same general pattern as before. This time, just as with the last simulation, the overall maximum value belongs to iteration 9. As with the previous force applications, the values for the displacement remain rather consistent with a drop off on the validation study performed with iteration 10. In this application, the values Safety Factors are fairly consistent, and do not show as much of a descending trend as with the previous force applications. This trend can be seen in **Figure 23**. For visualization purposes, we have provided a comparison of the displacement gradients produced by the outputs of the three simulations on our baseline model in **Figure 24**. These are

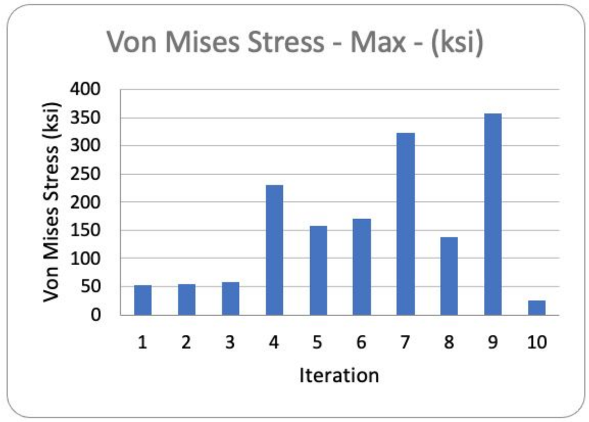


Figure 22: Two Pressures Von Mises Max

Figure 23: Two Pressures Safety Factor Min

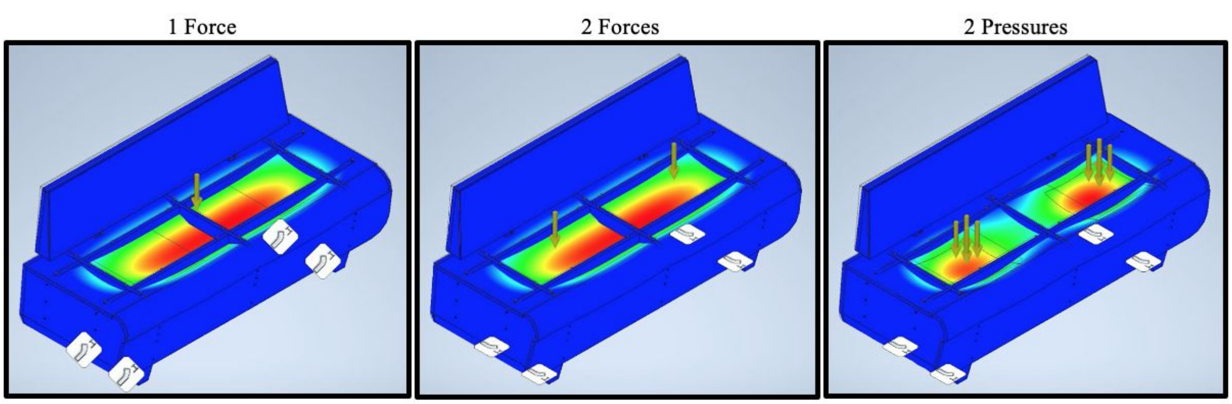


Figure 24: Comparison of Displacements

the only resulting images that have a visible gradient, as the other outputs produced more specifically localized gradients, which we will cover in the next chapter.

Chapter 8: Conclusions

As mentioned in the Chapter 2, it is important to note that "the intention of performing FEA is not to eliminate the real tests but rather to reduce the time for a product to pass through the process" (Rundgren and Wörmke, 2011). This means that the conclusions provided are a stepping off point for more research and real-life testing.

8.1 - Interpretation of Results

Based on the FEA conducted, the best option for the client is to remain with the current bench model, or the Bolted (Baseline) model. This model has the lightest weight that has a Safety Factor above one for the One Force and Two Force application methods. For the Two Pressure application method, its Safety Factor is just below one, suggesting there might be slight plastic deformation at one of the screws at the front of the base of the bench. **Figure 25** shows four models above the safety factor threshold, the Extra Support, the Original (Solid Feet), the Bolted (Baseline), and the Plywood models. However, the company is already having logistical issues with workers loading and unloading the bench at its current weight, so while adding support like in the Extra Support model would help make the bench more capable of handling heavy loads, it would also be infeasible and counter productive with the existing problems related to and the goal of reduction of the weight.

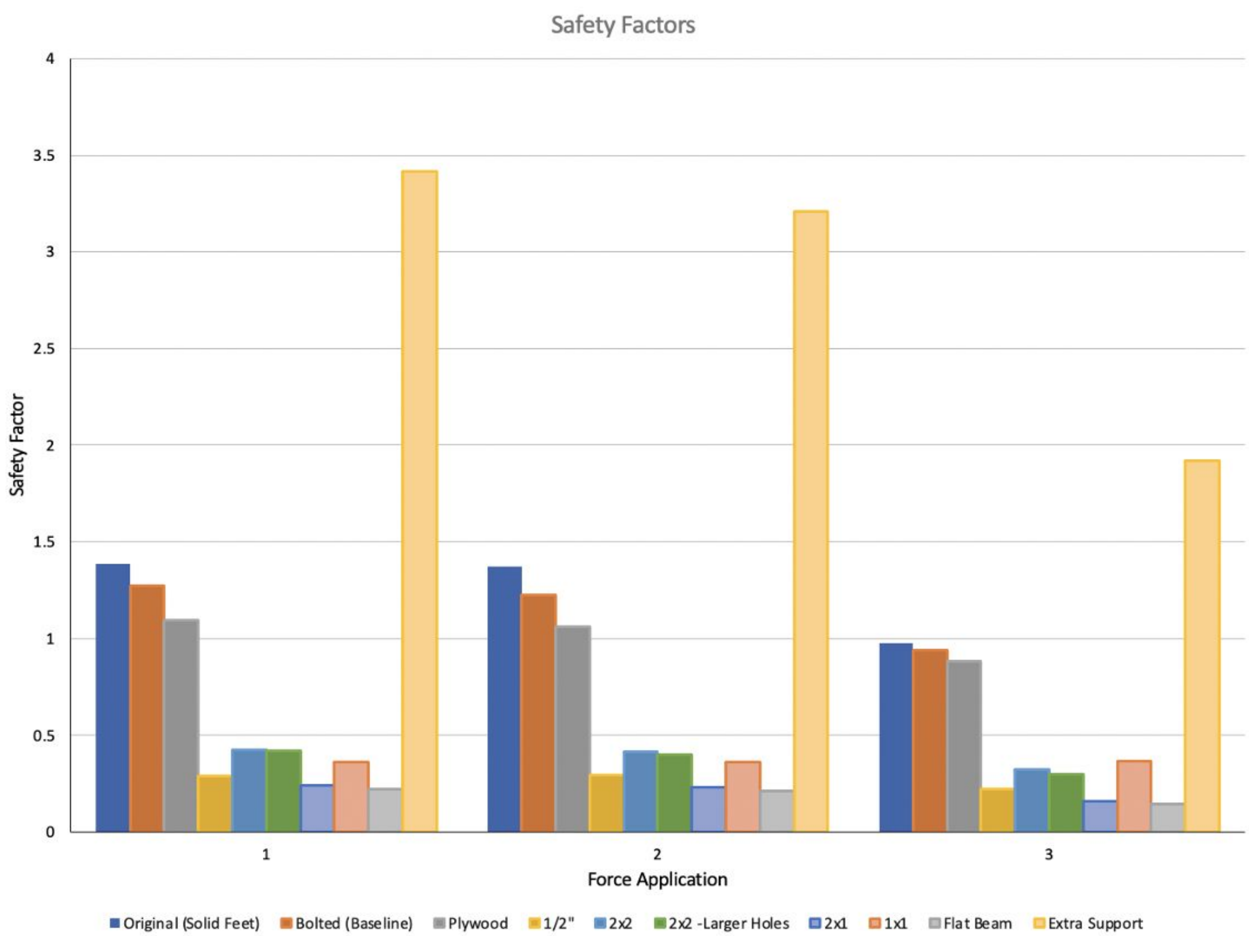


Figure 25: Safety Factor Summary

The majority of the safety factors produced by our analyses were too low for production due to an abrupt drop at a specific location on the model. We found that the minimum safety factor was located on several screws while the remainder of the model remained at the maximum value. **Figure 26** provides a zoomed image of the minimum safety factor gradient located on a screw from our baseline model, and **Figure 27** shows the entirety of that model and the safety factor gradient with the vast majority at the maximum value.

If the company decides they would like to come up with a cheaper alternative to the existing design at the expense of a one pound increase of weight, they could test a bench design with all of the Russian Birch wood replaced with a wood composite such as Plywood. As covered in the Literature Review, wood composites are shown to have a similar strength to solid wood (Kasal, 2006). This was confirmed in our FEA, where the minimum Safety Factor for the three applications of stress for the Plywood model are comparable, albeit slightly lower, than the Bolted (Baseline) model. Based on the results of the FEA, the next step could be real-life strength testing of a bench made out of wood composites.

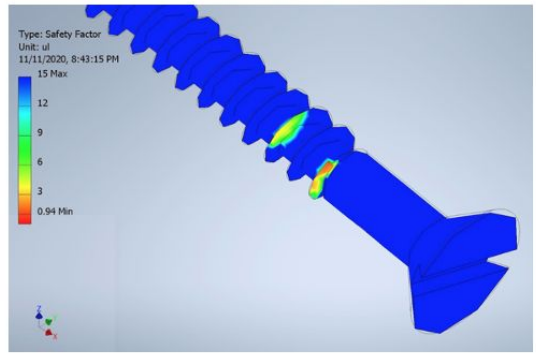


Figure 26: Screw Safety Factor Gradient

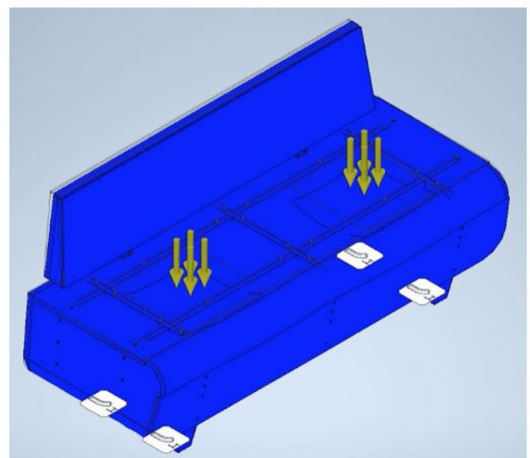


Figure 27: Bench Safety Factor Gradient

8.2 - Progress Toward Goals

Looking back at the group's intentions of the start of the semester, one of the three potential areas of improving the client's daily work processes was successfully researched and the company has been advised with next steps. While we initially had several potential areas of investigation, ranging from adding an overhead crane to conducting a warehouse optimization, we realized in discussions with the company that weight reduction of the bench would do the most to ameliorate their problems. While the other initial proposed solutions may not have been part of our methodology, their inclusion in the Literature Review helps communicate other potential ideas for further improvement to the company besides just reducing weight of the bench. Plus, the Finite Element Analysis we conducted addresses a top priority for the company that has been communicated as a goal for them from the start.

8.3 - Recommendations for Future Studies

Our team has a few recommendations for fleshing out this study. First, we would suggest using multiple FEA softwares, including SolidWorks and ANSYS, to verify results across platforms and improve validity. Also, we would recommend experimenting with more materials after collecting accurate physical properties via real-life testing. Birch and Plywood were a good start, but examining other wood and wood composites could help with saving money in manufacturing costs and avoiding over-dimensioning. Lastly, we would recommend testing the same alternatives we did to verify results while adding more alternatives, both with more stability and less stability, in order to fully explore all potential options and move on to real life testing for the few that have the best results from the FEA.

References

Azadeh, A. (. 1.)., et al. “Optimization of Facility Layout Design with Ambiguity by an Efficient Fuzzy Multivariate Approach.” International Journal of Advanced Manufacturing Technology, vol. 84, no. 1–4, pp. 565–579. EBSCOhost, doi:10.1007/s00170-015-7714-x. Accessed 13 Sept. 2020.

Chen, Yushu, and Zhihui Wu. “Study on Structure Optimization Design of Modified Wood Furniture Tenon Structure Based on the Finite Element Analysis of ANSYS.” Journal of Intelligent & Fuzzy Systems, vol. 34, no. 2, Feb. 2018, pp. 913–922. EBSCOhost, doi:10.3233/JIFS-169385.

Chowdary, Boppana V, et al. “Redesign of a Furniture Industry Component: A Sustainable Design Approach.” The West Indian Journal of Engineering, vol. 41, no. 2, 18 Jan. 2019, pp. 77–83., sta.uwi.edu/eng/wije/documents/WIJEv41n2Jan2019Completed.pdf#page=79.

Güray, Ersan & Kasal, Ali & Erdil, Yusuf & Kuşkun, Tolga & Acar, Mehmet. (2015). Determination of the Minimum Weight of Furniture Frames with Subjected to Required Strength Constraints.

Kasal, Ali. “Determination of the Strength of Various Sofa Frames with Finite Element Analysis.” Gazi University Journal of Science, vol. 19, no. 4, Oct. 2006, pp. 191–203.

EBSCOhost, search.ebscohost.com/login.aspx?direct=true&AuthType=ip,shib&db=a9h&AN=23250672&site=eds-live&scope=site.

Klemperer, Justus, et al. "Careful with that warehouse: the furniture business requires movers and shakers who know how to optimize." Industrial Engineer, vol. 35, no. 9, Sept. 2003, pp. 40-46. Gale Academic OneFile, https://link.gale.com/apps/doc/A112818678/AONE?u=kennesaw_main&sid=AONE&xid=7704f2af. Accessed 13 Sept. 2020.

Koç, K. Hüseyin, et al. “The Use of Finite Element Method in the Furniture Industry.” African Journal of Business Management, vol. 5, no. 3, 4 Feb. 2011, pp. 855–865., doi:10.5897/AJBM10.551.

Lipinskis, Inokentijs, and Uldis Spulle. “Research On Mechanical Properties Of Birch Plywood With Special Veneer Lay-Up Schemes.” Drewno, vol. 54, no. 185, 2011, pp. 109–118.

Lunchbox. “CAT Flat Washer.” GrabCAD Community, 22 Jan. 2012.

Lunchbox. “CAT Hex Head Bolt- Metric.” GrabCAD Community, 22 Jan. 2012.

Mansfield, Graham. “031.00.285 M8.” GrabCAD Community, 5 June 2012.

Min Zhang, et al. “Modeling of Workflow Congestion and Optimization of Flow Routing in a Manufacturing/Warehouse Facility.” Management Science, vol. 55, no. 2, 2009, p. 267. EBSCOhost, doi:10.1287/mnsc.1080.0916.

Nyemba, Wilson R. & Mbohwa, Charles. "Process Mapping and Optimization of the Process Flows of a Furniture Manufacturing Company in Zimbabwe Using Machine Distance Matrices." Procedia Manufacturing, vol. 8, Dec. 2017, pp. 447-454. Science Direct, doi: 10.1016/j.promfg.2017.02.057. Accessed 13 Sept. 2020.

Petitt, Josh. “SCRW_WSFH 06-22-06.” Autodesk, 22 June 2006.

Pfeifer, Michael. Materials Engineering Selection: a Product Life Cycle Approach. Butterworth-Heinemann, 2009. ScienceDirect, doi.org/10.1016/B978-0-7506-8287-9.00006-9.

Rundgren, Anders, and Fredrik Wörmke. Optimisation of the Ektorp Sofa Frame through the Use of Finite Element Analysis. 2011, http://urn.kb.se/resolve?urn=urn:nbn:se:mdh:diva-12466.

“Safety Factor.” Autodesk Support & Learning, Autodesk, 27 Aug. 2014, knowledge.autodesk.com/support/inventor-products/learn-explore/caas/CloudHelp/cloud help/2015/ENU/Inventor-Help/files/GUID-D541AE00-F9FB-4B7E-A595-402219353010-htm.html.

Sanchez, Juan, et al. "Improving the Stainless Steel System at Process Technology International Using 5S, Process Improvement, and Inventory Control." 22 Apr. 2020, pp. 1–68. Accessed 13 Sep. 2020.

Siemiatkowski, Mieczyslaw S., and Maria Vargovska. "Process Layout Planning and Optimised Product Range Selection in Manufacture of Wooden Construction Sets." Maderas: Ciencia y Tecnología, vol. 21, no. 2, Apr. 2019, pp. 171–184. EBSCOhost, doi:10.4067/S0718-221X2019005000205.

Tankut, Nurgul, et al. "Finite Element Analysis of Wood Materials." Drvna Industrija, vol. 65, no. 2, 9 Apr. 2014, pp. 159–171., doi:10.5552/drind.2014.1254.

Wengang HU, et al. "Optimal Design of a Furniture Frame by Reducing the Volume of Wood." Drewno, vol. 62, no. 204, Apr. 2019, pp. 85–97. EBSCOhost, doi:10.12841/wood.1644-3985.275.12.

Yifei Zhao, and Stein W. Wallace. "Integrated Facility Layout Design and Flow Assignment Problem Under Uncertainty." INFORMS Journal on Computing, vol. 26, no. 4, Fall 2014, pp. 798–808. EBSCOhost, doi:10.1287/ijoc.2014.0599.

Appendix A: Acknowledgements

We would like to Acknowledge the following for their support of our project.

- Will Mack - VP of Special Projects
- Dennis Batin - Director of Engineering
- All employees of CBBE
- Gregory Wiles - ISYE Department Head
- The Kennesaw State Industrial Engineering Department
- Professor Adeel Khalid - Senior Design Professor
- The Kennesaw State Mechanical Engineering Department
- Professor Santana Roberts - Senior Design Professor
- Adam Ricketts - Mechatronics Engineering Student

Appendix B: Contact Information

Garrett Smith: smithgarrett381@gmail.com

Ernie Rivera: ernspeare@gmail.com

Austin Hester: austin_hester@bellsouth.net

Jake Santa Cruz: santacruz.jake@gmail.com

Dr. Adeel Khalid: akhalid2@kennesaw.edu

Will Mack, VP of Special Projects, CBBE: wmack@cbbe.net

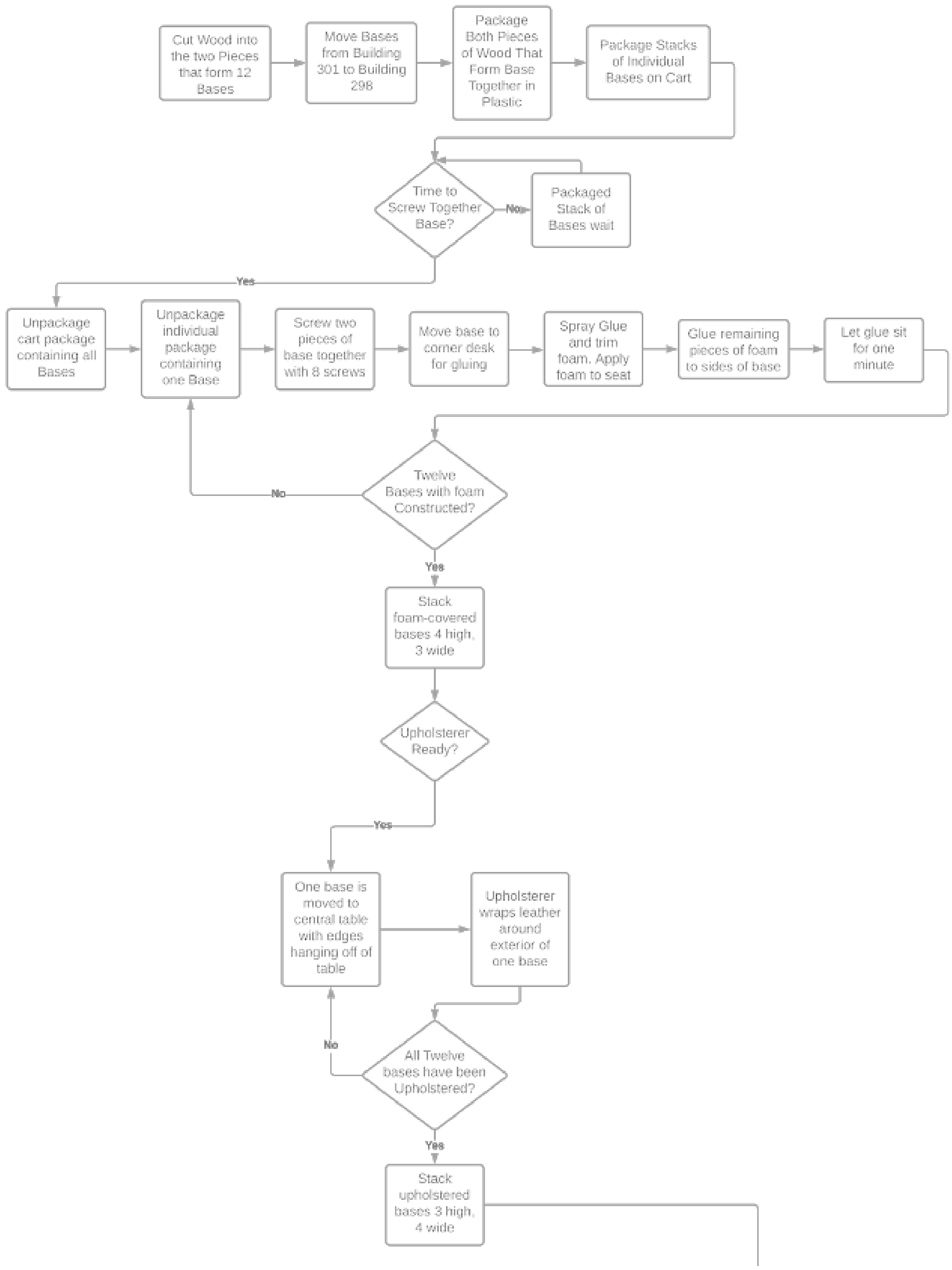
Denis Batin, Director of Engineering, CBBE: dbatin@cbbe.net

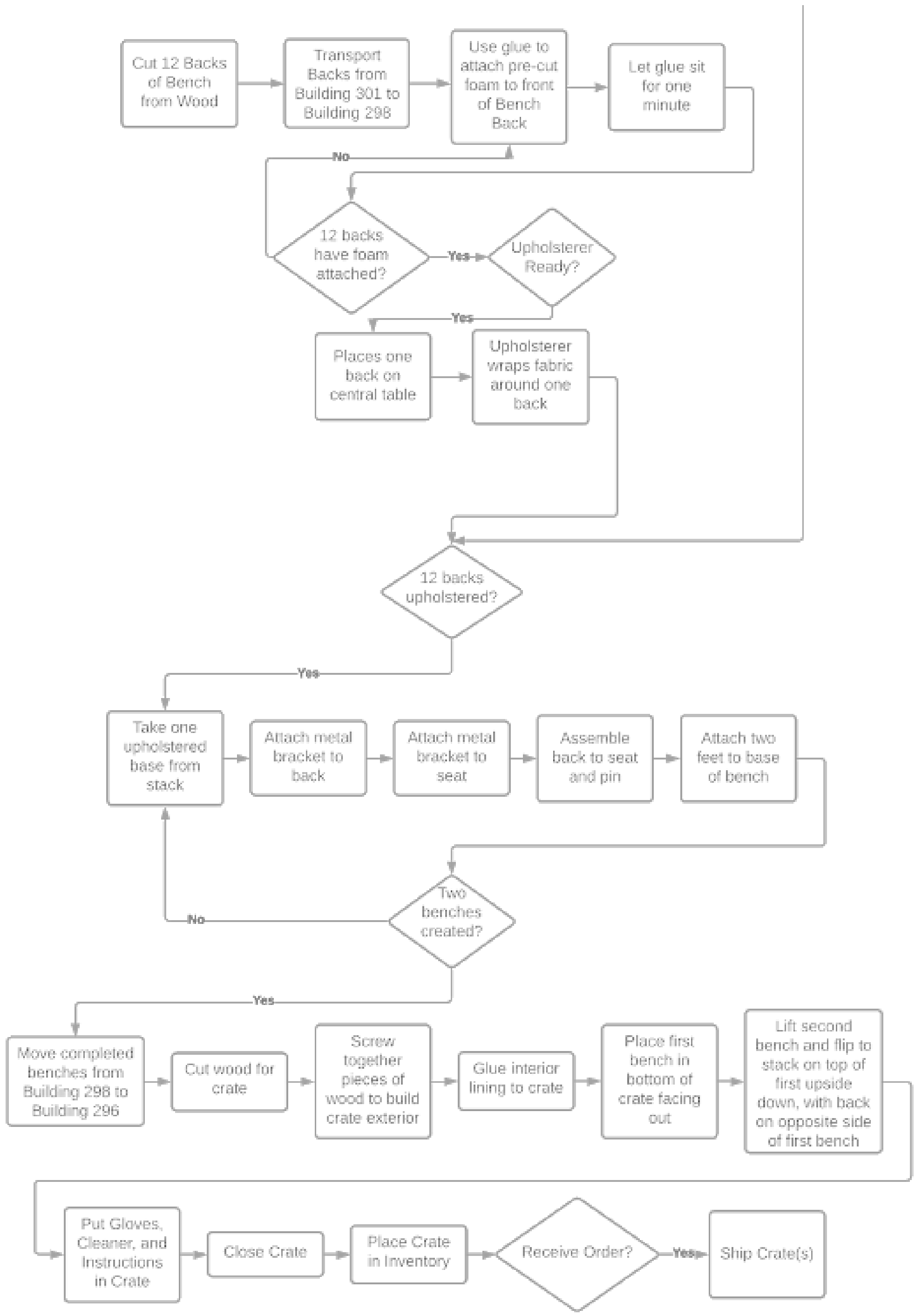
Santana Roberts, Mechanical Engineering Professor: srobe157@kennesaw.edu

Appendix C: Reflections

This project has taught us many different things including how to work and manage teams remotely. Due to the CoronaVirus pandemic, our project was forced into an online capacity, necessitating our group to perform numerous virtual meetings and greatly limited our ability to perform on-site visits. Our project scope moved away from the general coursework provided to Industrial and Systems Engineering students, and we were required to conduct large amounts of learning about the FEA process and the softwares used to perform them. To overcome this obstacle, we consulted with students with experience performing such analyses as well as consulting with professors within the Mechanical Engineering department. We had little choice in what software we used to perform our analyses, since the Solidworks FEA package exceeded our budget. Inventor became the most viable option due to the monetary savings provided, the ease of transfer from the provided models, and our familiarity to the tools and software itself.

Appendix D: System Block Diagram



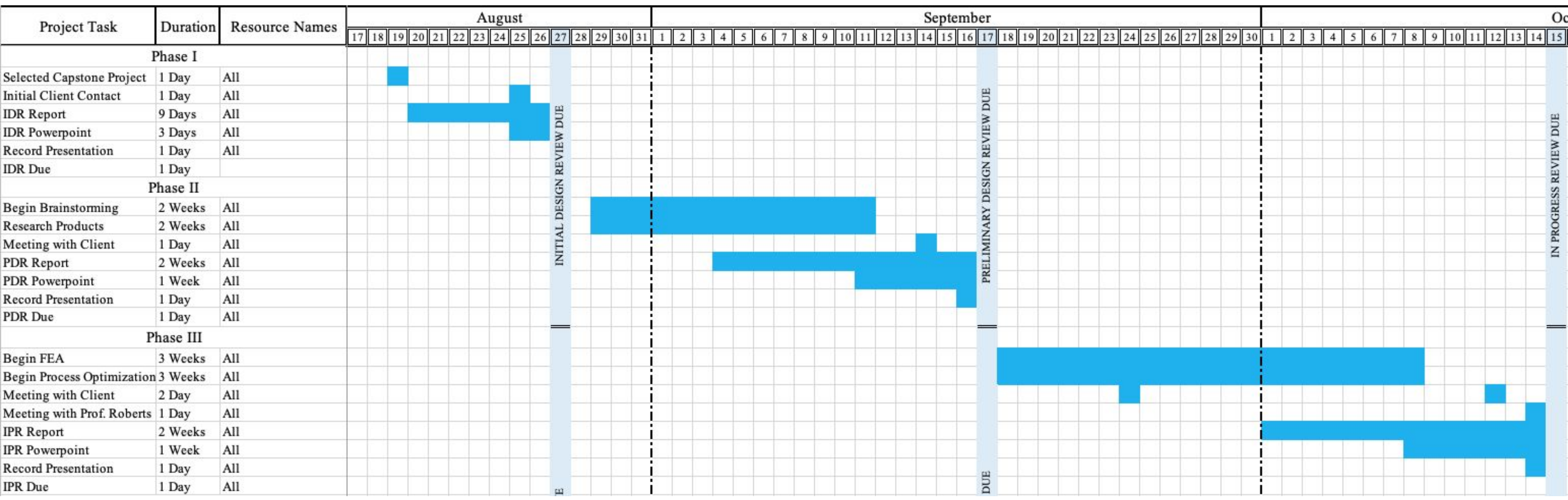

Cut 12 Backs of Bench from Wood
Transport Backs from Building 301 to Building 298
Use glue to attach pre-cut foam to front of Bench Back
Let glue sit for one minute
No
12 backs have foam attached?
Yes
Upholsterer Ready?
Yes
Places one back on central table
Upholsterer wraps fabric around one back
12 backs upholstered?
Yes
Take one upholstered base from stack
Attach metal bracket to back
Attach metal bracket to seat
Assemble back to seat and pin
Attach two feet to base of bench
No
Two benches created?
Yes
Move completed benches from Building 298 to Building 296
Cut wood for crate
Screw together pieces of wood to build crate exterior
Glue interior lining to crate
Place first bench in bottom of crate facing out
Lift second bench and flip to stack on top of first upside down, with back on opposite side of first bench
Put Gloves, Cleaner, and Instructions in Crate
Close Crate
Place Crate in Inventory
Receive Order?
Yes
Ship Crate(s)

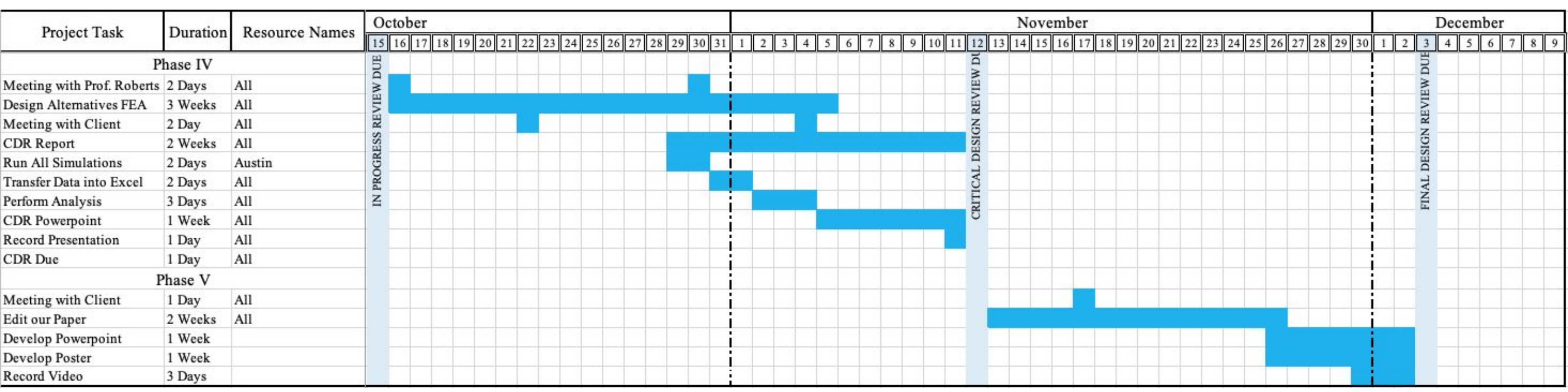
Appendix E: Engineer Budget

	Budget						
Project Name		CBBE Bench v2 Optimization					
Last Update		11/10/2020					
						Total Cost	$113,233
Materials:							
Crane		$	50,000.00				
Glue Cart		$	200.00				
FEA Student Software							
----Autocad		$	2,720.00				
----InCAD DesignPak (ALGOR)		$	975.00				
----DesignSpace (ANSYS)		$	4,000.00				
----FEMAP (SDRC)		$	3,000.00				
----Working Model FEA (MSC)		$	6,000.00				
----COSMOS (SRAC)		$	6,000.00				
----Nastran (SimEvolution)		$	8,588.00				
		$	81,483.00				

Manpower:						
	Estimated work hours by task				**Totals by Task**	
Task	**Garrett**	**Ernie**	**Jake**	**Austin**	**hrs**	**$**
IDR	20	20	20	20	80	$ 4,500.00
PDR	30	30	30	30	120	$ 6,750.00
IPR	30	45	40	50	165	$ 9,000.00
CDR	30	25	30	40	125	$ 7,000.00
FDR	20	20	20	20	80	$ 4,500.00
					0	$ -
					0	$ -
Total hrs	130	140	140	160	570	
Rate $/hr	75	50	50	50		
Dollars	$9,750	$7,000	$7,000	$8,000		$31,750

Appendix F: Gantt Chart





Appendix G: Group Contributions

Team Member Contributions				
Name	Garrett Smith	Ernie Rivera	Austin Hester	Jake Santa Cruz
Position	Project Manager, Process Engineer	Project Coordinator, Systems Engineer	Technical Expert, Operations Research Engineer	Data Analyst, Quality Engineer
Chapter 1	✓			✓
Chapter 2		✓		✓
Chapter 3	✓	✓		
Chapter 4	✓	✓		
Chapter 5	✓		✓	
Chapter 6		✓	✓	
Chapter 7			✓	✓
Chapter 8			✓	✓
Major Contributions:	Project management, point of contact with client	Final Poster, Gantt Chart, prototype testing	Developed models, ran simulations, analyzed results, created graphics	System Block Diagram, Final Video Demonstration, Literature Review