CISCO NEXUS 9200 ROBUSTNESS REDESIGN

A Senior Project submitted to
the Faculty of California Polytechnic State University,
San Luis Obispo

In Partial Fulfillment
of the Requirements for the Degree of
Bachelor of Science in Industrial & Manufacturing Engineering

by
Ethan Gold, Colin Berge, Cole Christophersen, Rahul Makhijani
June 14, 2019

ABSTRACT

Cisco Nexus 9200 Robustness Redesign

Ethan Gold, Colin Berge, Cole Christopherson, Rahul Makhijani

Certain configurations of Cisco's Nexus 9200 product experienced issues with bending during shipping. Two solutions were developed to eliminate this problem: an external brace that could quickly address the problem yet was expensive and unsustainable, and a redesigned chassis, which was more economical but came with a longer time to implement. Real world packaging and shipping conditions were simulated in SolidWorks and Finite Element Analysis was used to model the stresses experienced when the product is dropped. Both designs were found to significantly reduce stress in critical areas, thus reducing the chance of failure and the cost of the problem. We recommend Cisco implement the external brace to fill the time until the redesigned top of the chassis is implemented, both of these will reduce failures and slash the cost of replacing failed units.

Figure 1: Short Term Solution, Nexus 9200 with External Brace Attached

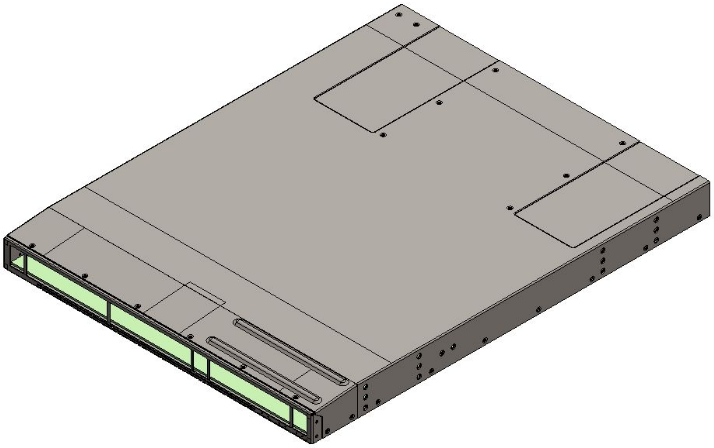


Figure 2: Long Term Solution, Nexus 9200 with Various Chassis Enforcements

ACKNOWLEDGMENTS

It is a great pleasure to thank our project sponsor Cisco Systems for providing our senior project team with a legitimate industry problem. Being given the opportunity to put our problem solving skills to use, reinforce our communication skills with respected members of their company and acquire new skills throughout this process was very rewarding.

First of all, we would like to thank our Cisco contact, Christine Nolan-Brady, an employee of the Test and Quality department. Without her continuous guidance, access to Cisco information, and connection to the teams that work on the Nexus 9200, our project would not have been completed in a manner that it was.

Second, our team way instructed by Dr. Tali Freed and Roy Jafari, Industrial & Manufacturing Professors at California Polytechnic State University. With their support and advising our team stayed on task and was able to meet deadlines.

Thirdly, there is credit due to the numerous Cal Poly professors that aided our team in solution design as well as Finite Element Analysis design and validation. These professors are Trian Georgeau, IME, who helped with solution designs an initial FEA simulation design. Dr. Peter Schuster, Mechanical Engineering, & Dr. Garett Hall, Civil Engineering, who aided our simulation design and validation methods.

Finally, Henry Lee, a Cisco expert in FEA, who informed our group on the use of FEA and also validated the design of our model.

These great people all helped our group learn, grow and provide Cisco with options for their future designs of the Nexus 9200.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

2. Introduction and Background

In 2018, Cisco started receiving reports that some Nexus 9200 switches were showing up to customer sites deformed. The deformation restricted customers ability to install the switches and open them up for maintenance. Cisco addressed the problem by reinforcing the top of the units, modifying the foam packaging, and creating a separate compartment in the box to store accessories, which were thought to contribute to the damage. Though this successfully eliminated the problem, the Cisco Test & Quality team reached out to the Cal Poly IME (Industrial and Manufacturing Engineering) department to get a second opinion in the form of a senior project.

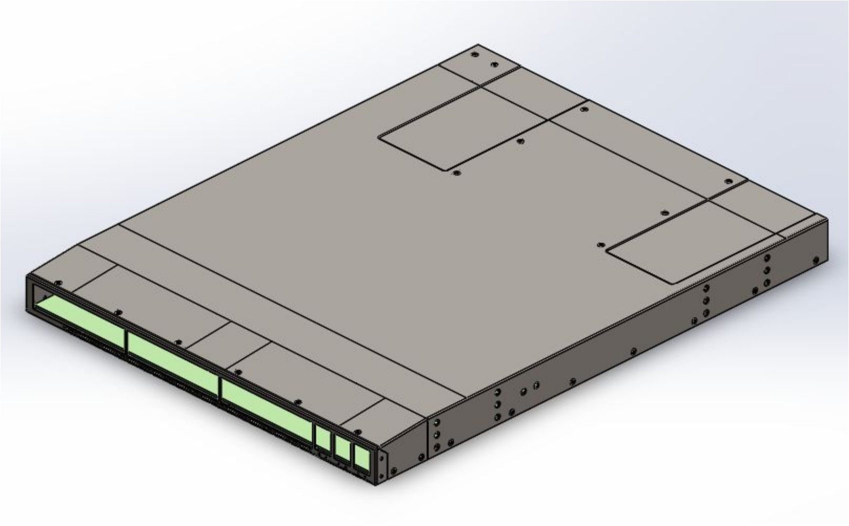


Figure 3: CAD Front view of a Nexus 9200

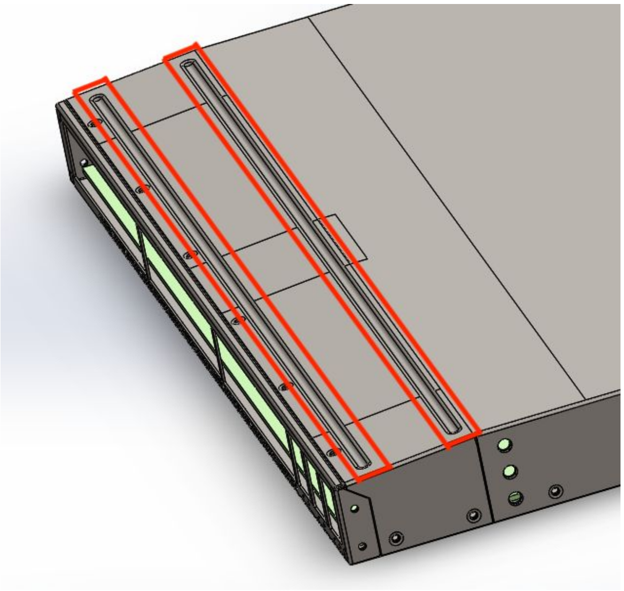


Figure 4: CAD View of Nexus 9200 with Top Cover Reinforcements

The Cal Poly team was asked to develop a solution for these products as well as develop guidelines for designing future products that would identify designs prone to failure and increase their durability. As changes to the products would take a long time to implement, the team decided to break our solution into two parts. The first was a short term solution, external to the product itself, that could be implemented quickly. The second was a long term, sustainable solution which would require product changes and thus take longer to implement.

The Nexus 9200 product line is a family of rack mounted switches. A switch is a computer networking device that receives, processes, and shares data between devices on a network. The "rack mounted" designation means it is specifically made to be housed in a server rack, a cabinet made to hold servers that provide power and internet capabilities. These products are typically installed in server racks on rails that allow the units to slide in and out easily for service.

When the problem existed, Cisco replaced each failed unit and shipped the damaged ones back for failure analysis by engineers. This meant that for each failure, Cisco was incurring the costs of manufacturing and shipping two units to the customer as well as reverse shipping the damaged one back. These failures resulted in an additional cost of approximately $X,XXX,XXX per year, or roughly $8.30 per unit sold.

3. Problem Description

This section describes the current state of the issue, discusses potential causes, and explains the economic impact of leaving the issue unresolved.

3.1 Current State

Despite the original design passing Cisco testing procedures, Nexus 9k chassis were becoming deformed during shipping. Specifically, the top and bottom sheet metal covers would become concave down, resulting in a chassis that appeared to look like it was “frowning.” The effects of the deformation ranged from aesthetic to impeding installation and maintenance.

An original, undamaged Nexus 9200 chassis appears as follows:

Figure 5: Front View of Physical Nexus 9200 Chassis

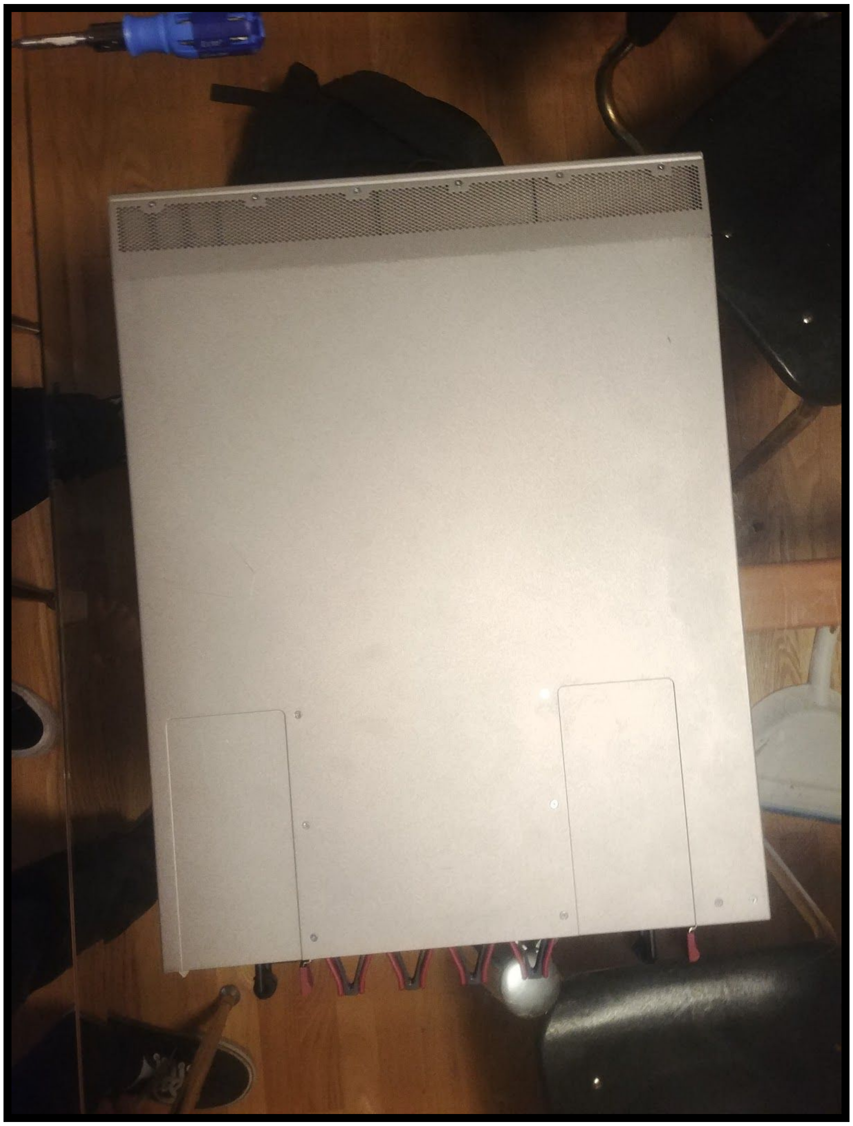


Figure 6: Top View of Physical Nexus 9200 Chassis

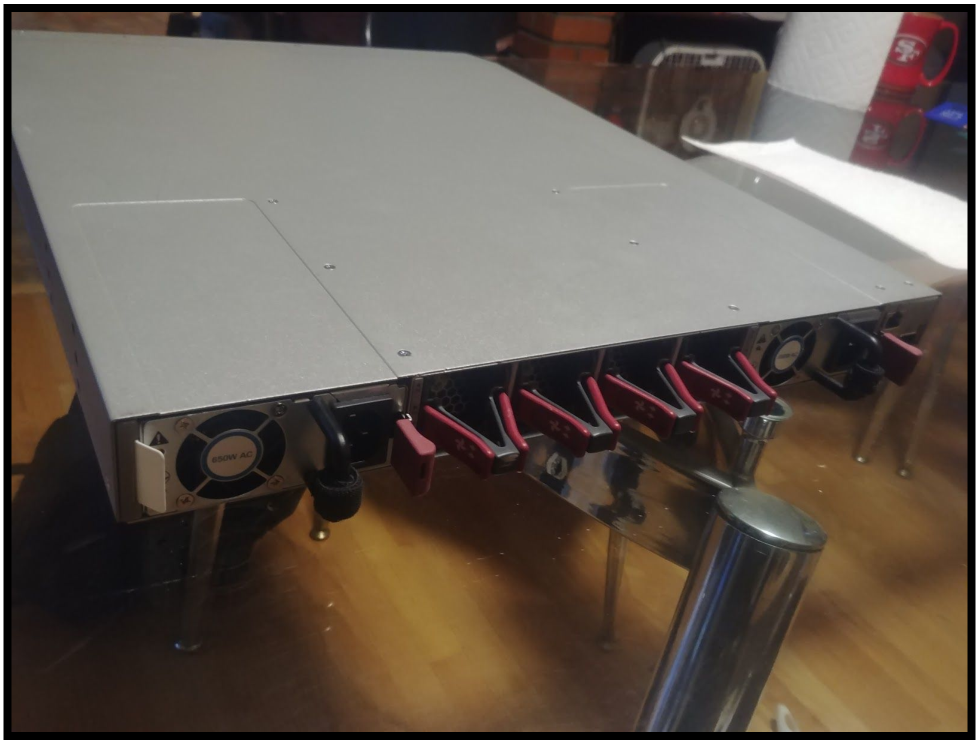


Figure 7: Rear View of Physical Nexus 9200 Chassis

On the contrary, an example of a chassis that was damaged during shipping can be found below. The white arrow on the image is pointing to the bent chassis.

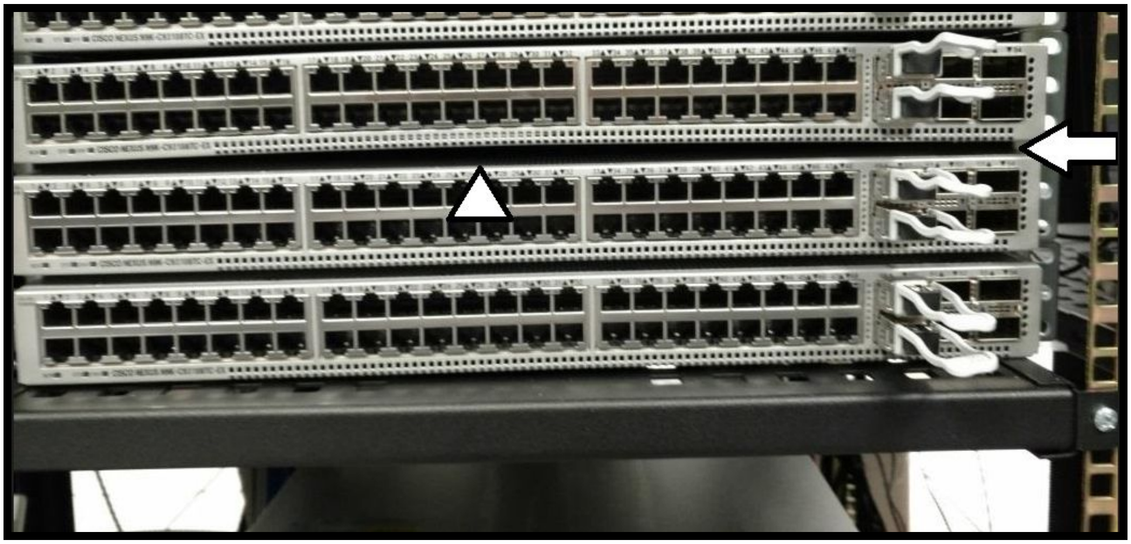


Figure 8: Front of Bent Nexus 9200 Chassis

Furthermore, an example of a chassis that experienced damage to the SFP ports on the front of the chassis can be found in Figure 7 below.

Figure 9: Nexus 9200 Chassis with Damaged Front Ports

Damage occurring to the SFP ports would prevent cables from plugging in, and thus instances such as these would prevent the chassis from being fully usable.

3.2 Potential Causes

The team analyzed the issue from three perspectives - product, packaging, and process - and after several meetings with our stakeholders, brainstormed potential reasons for the failures and put them into a fishbone diagram.

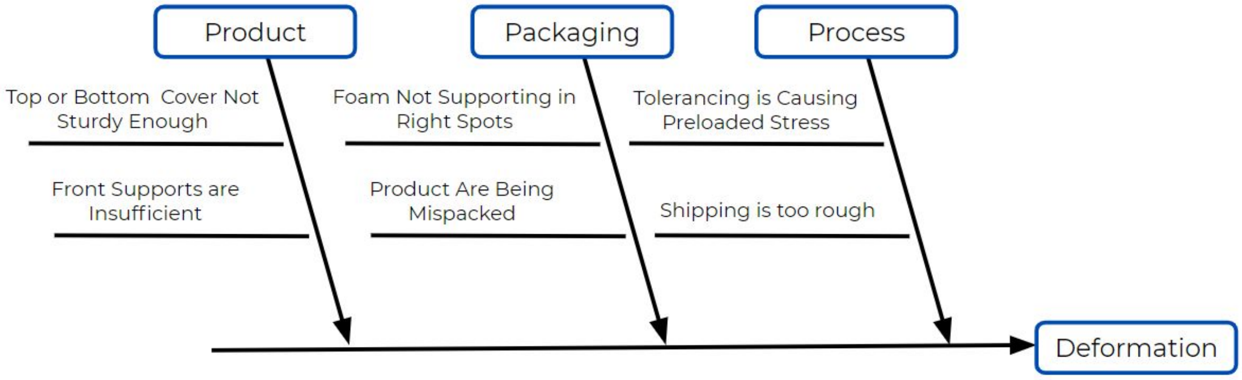


Figure 10: Fishbone Diagram of Potential Causes of Deformation

From a product standpoint, the top and bottom metal covers were not strong enough to withstand stress garnered from shipping. The spacing and design of the front supports caused them to overload and buckle during shipping.

From a packaging perspective, it was possible that the foam was not supporting the chassis in the appropriate way. Cisco conducted a case study and discovered that sometimes the packagers of the units were inappropriately placing accessory items (cables, power supplies, etc.) directly on top of the chassis rather than in the designated slots. These items were hitting the chassis during shipping and likely contributed to bending.

Finally, from a process perspective, there was a possibility that the tolerances of the sheet metals were adding a preloaded stress to the top cover upon assembly. This could have been happening when the space between the top and bottom covers was at a maximum based on their acceptable tolerances and the metals were being forced together with screws.

Cisco saw its deformed units with a select number of buyers who were using third party logistics that were shipping individual units at a time. There are many factors that went into if a unit deformed or not, but it was acknowledged that these factors were only an issue in the most extreme circumstances. For this reason, a root cause analysis was outside the scope of this project and the team focused solely on product redesigns to increase the robustness of the chassis.

3.3 Economic Evaluation of Current State

A flow diagram of the process that occurred for each damaged unit, as well as the costs associated with it, is shown in Figure 11 below. Processes are shown in blue and costs are shown in red.

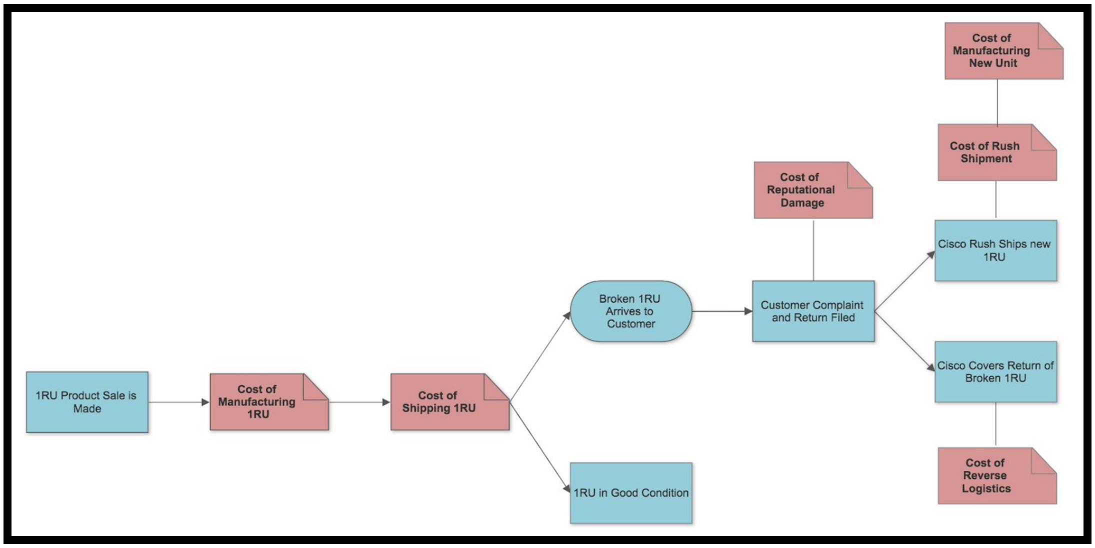
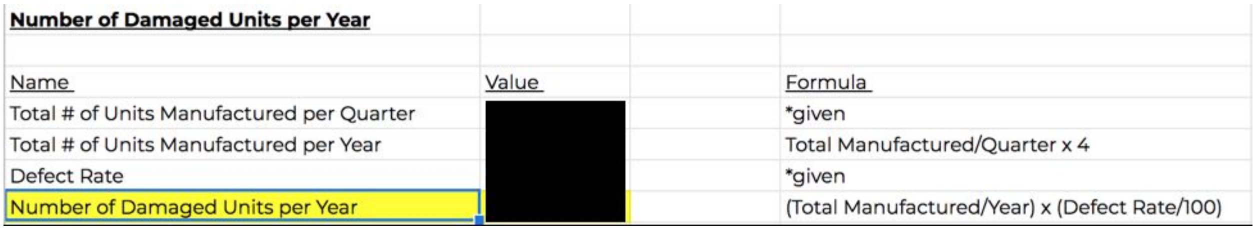


Figure 11: Process Due to Damaged Unit Including Costs Incurred

Cisco incurred four additional costs with every damaged chassis. There was the cost of manufacturing a new unit, rush shipping the customer the new unit, reverse logistics for the damaged unit, and reputational damage to its noteworthy name and brand. Reputational damage can open up avenues for competitors to capitalize with a powerful go-to-market strategy and competitive advantage.

At a production rate of XXX,XXX units per year, and an estimated defect rate of X% (provided by Cisco), an estimated XXX Nexus 9200's per year were experiencing deformation (Table 1).



Number of Damaged Units per Year

Name	Value		Formula
Total # of Units Manufactured per Quarter	[redacted]		*given
Total # of Units Manufactured per Year	[redacted]		Total Manufactured/Quarter x 4
Defect Rate	[redacted]		*given
Number of Damaged Units per Year	[redacted]		(Total Manufactured/Year) x (Defect Rate/100)

Table 1: Number of Damaged Nexus 9200s per year

Accounting for the associated costs with every damaged unit, the estimated total cost of the problem was $X,XXX,XXX per year and increased the overall unit cost by $8.30 per unit (Table 2).

Cost of Overall Problem

Name	Value		Formula
Number of Damaged Units per Year	[redacted]		*see previous table
Fully Allocated System Cost of one 1RU	[redacted]		*given
Shipping Cost of one 1RU from CM, DF	[redacted]		*given
Cost of Reverse Logistics (one 1RU back to Cisco)	[redacted]		*given
Total COGS to Cisco per Damaged Unit	[redacted]		System Cost + Shipping Cost + Reverse Logistical Cost
Total Cost of Overall Problem	[redacted]		Total Cost to Cisco/Unit x Number of Damaged Units per Year

Table 2: Yearly Cost of Nexus 9200 Damage

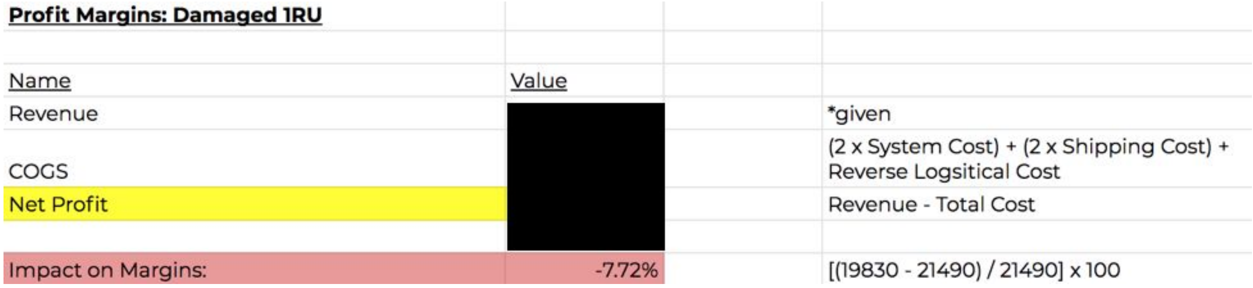
Another calculation of interest was the effect that each damaged 1RU unit had on the profit margins associated with the sale. According to Cisco, the average sale price of each 1RU unit was approximately $XX,XXX. Given that the cost of manufacturing and shipping each 1RU was approximately $X,XXX, the profit generated from each undamaged 1RU was approximately $XX,XXX. It should be noted that this estimate fails to include costs associated with direct labor and overhead which would drastically decrease profit margins. These numbers were unobtained due to limited access to factory and company information. Regardless, the calculation for the overall estimate can be found in Table 3 below.

Profit Margins: Regular 1RU

Name	Value		Formula
Revenue	[redacted]		*given
COGS	[redacted]		System Cost + Shipping Cost
Net Profit	[redacted]		Revenue - Total Cost

Table 3: Profit Margins Regular 1RU

On the contrary, the total COGS associated with each damaged 1RU increased to $X,XXX. This cut the net profit generated from each damaged 1RU down to $XX,XXX and thus reduced the total profit margin by -7.72%. The calculation for that estimate can be in Table 4 below.



Profit Margins: Damaged 1RU

Name	Value		
Revenue	[redacted]		*given
COGS	[redacted]		(2 x System Cost) + (2 x Shipping Cost) + Reverse Logsitical Cost
Net Profit	[redacted]		Revenue - Total Cost
Impact on Margins:	-7.72%		[(19830 - 21490) / 21490] x 100

Table 4: Impact on Profit Margin - Damaged 1RU

Again, as previously mentioned, these financial estimates do not include costs associated with overhead and direct labor. However, it should also be noted that the -7.72% impact on profit margins is a best case scenario; factoring in increased costs of direct labor and overhead would cut margins even further.

4. Literature Review

This section is composed of reports put out by Cisco and textbooks relevant to the packaging and simulation method used within our project. Also contained in this section is the information and guidance received from three Cal Poly, San Luis Obispo professors and Henry Lee of Cisco.

4.1 Literature Review

The literature reviewed was a case study performed by Cisco, package test reports done by Cisco, textbooks on packaging dynamics and stress, strain calculations and finally a lecture on the Finite Element Method.

4.1.1 Root Cause Analysis

In order to establish practical design changes our team needed to know about the current design of Cisco's Nexus 9200. In January 2019, a case study was performed to investigate the root cause of the bent chassis and damaged SFP ports. Cisco discovered that improper packaging was the root of this cause of bending. By placing accessories on the top cover of the chassis and dropping the package from an increased drop height, 36 inches, Cisco was able to replicate the bending damage seen by customers. To solve this issue Cisco decided to prevent this issue by adding additional packaging space for the accessories (Figure 12). Drop testing the new packaging, with accessories placed off of the chassis (green), prevented the bending damage.

Figure 12: Current Packaging

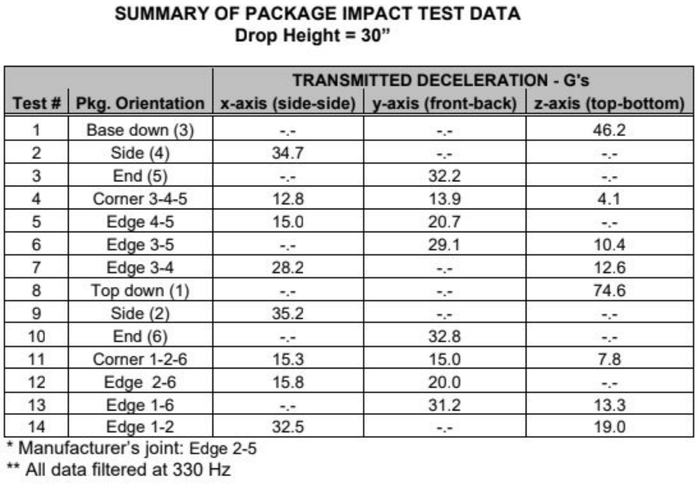
Cisco investigated the SFP port damage by measuring the durability of chassis surrounding the SFP ports. It was observed that the chassis surrounding the SFP ports can only withstand a 30 lbs force before deflection, compared to the 80lbs the area surrounding the QSFP ports. Two horizontal beads, above the SFP ports, were added in an attempt to increase the chassis durability. Our focus will be strengthening the areas of maximum stress in top cover and sheet metal surrounding SFP ports.

Figure 13: Current Top Cover with Beads

4.1.2 Packaging Dynamics & Current State

Understanding the current packaging used to protect the Nexus 9200 is vital to understanding the quantity of force the product will need to be capable of withstanding. Packaging test reports were shared by Cisco detailing vibration and drop testing on the Nexus 9200 chassis. Their purpose was to inspect their current packaging's ability to protect their product from physical damage. The packaging passed both drop and vibration tests. These reports provided insight as what type of drop results in the greatest force acting on the product.

In test 1 and 8 of both designs we observe the two maximum accelerations, in G's, exerted on the chassis comes from drops on the top and bottom of the packaging. The chassis was dropped from a height of 36 inches and experienced a force of 57.6 & 46.2 G's when upright and 60.4 G's & 74.6 G's upside down (Tables 5 and 6).

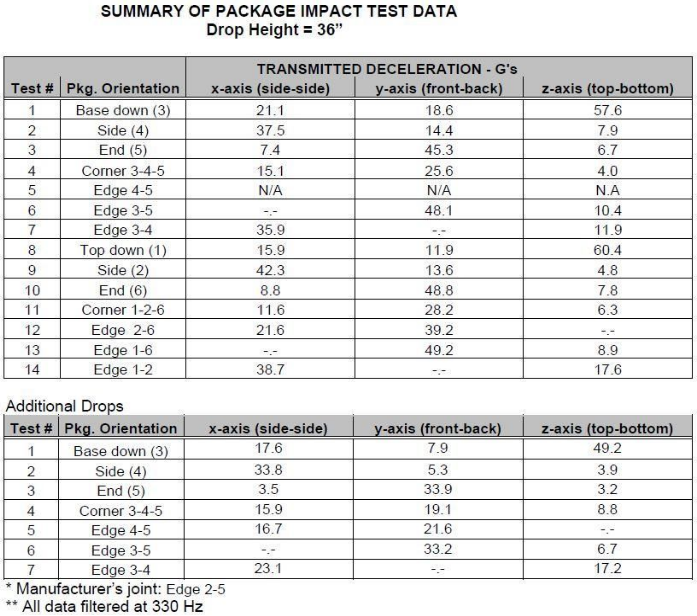


SUMMARY OF PACKAGE IMPACT TEST DATA
Drop Height = 30"

		TRANSMITTED DECELERATION - G's		
Test #	Pkg. Orientation	x-axis (side-side)	y-axis (front-back)	z-axis (top-bottom)
1	Base down (3)	-.-	-.-	46.2
2	Side (4)	34.7	-.-	-.-
3	End (5)	-.-	32.2	-.-
4	Corner 3-4-5	12.8	13.9	4.1
5	Edge 4-5	15.0	20.7	-.-
6	Edge 3-5	-.-	29.1	10.4
7	Edge 3-4	28.2	-.-	12.6
8	Top down (1)	-.-	-.-	74.6
9	Side (2)	35.2	-.-	-.-
10	End (6)	-.-	32.8	-.-
11	Corner 1-2-6	15.3	15.0	7.8
12	Edge 2-6	15.8	20.0	-.-
13	Edge 1-6	-.-	31.2	13.3
14	Edge 1-2	32.5	-.-	19.0

* Manufacturer's joint: Edge 2-5
** All data filtered at 330 Hz

Table 5: Package Test Data - 30 Inch Drop



SUMMARY OF PACKAGE IMPACT TEST DATA
Drop Height = 36"

		TRANSMITTED DECELERATION - G's		
Test #	**Pkg. Orientation**	**x-axis (side-side)**	**y-axis (front-back)**	**z-axis (top-bottom)**
1	Base down (3)	21.1	18.6	57.6
2	Side (4)	37.5	14.4	7.9
3	End (5)	7.4	45.3	6.7
4	Corner 3-4-5	15.1	25.6	4.0
5	Edge 4-5	N/A	N/A	N.A
6	Edge 3-5	-.-	48.1	10.4
7	Edge 3-4	35.9	-.-	11.9
8	Top down (1)	15.9	11.9	60.4
9	Side (2)	42.3	13.6	4.8
10	End (6)	8.8	48.8	7.8
11	Corner 1-2-6	11.6	28.2	6.3
12	Edge 2-6	21.6	39.2	-.-
13	Edge 1-6	-.-	49.2	8.9
14	Edge 1-2	38.7	-.-	17.6

Additional Drops

Test #	**Pkg. Orientation**	**x-axis (side-side)**	**y-axis (front-back)**	**z-axis (top-bottom)**
1	Base down (3)	17.6	7.9	49.2
2	Side (4)	33.8	5.3	3.9
3	End (5)	3.5	33.9	3.2
4	Corner 3-4-5	15.9	19.1	8.8
5	Edge 4-5	16.7	21.6	-.-
6	Edge 3-5	-.-	33.2	6.7
7	Edge 3-4	23.1	-.-	17.2

* Manufacturer's joint: Edge 2-5
** All data filtered at 330 Hz

Table 6: Package Test Data - 36 Inch Drop

Measurements were taken by an accelerometer placed in the middle, left side of the chassis. These numbers will be used to set a baseline for the minimum robustness that a chassis needs to be in order to be considered acceptable. The proposed design solution will have to be drop tested and record lower numbers to the top and bottom covers in order to verify our simulation.

To further understand how a drop test is run and the units of measure (G's) we reviewed <u>Protective Packaging for Distribution: Design and Development</u>, by Daniel Goodwin and Dennis Young. Testing for drops can be done using a Precision Drop Tester (PDT) using ASTM D5276 Standards. Based on package weight and assurance level drop height is determined. Standard drop height is 36 inches. Orientation is also key to a drop test. All sides, edges and corners are eligible to be dropped on. Selection of these is based on the area of interest. Our areas of interest are the top and bottom of the package. The motion the package and product experiences during a drop is shock. Shocks can occur anytime during loading, unloading, sorting and shipping. Shock is measured as a vector quantity G (G = acceleration/deceleration/force of gravity). Fragility is defined as the minimum G-Force the product that causes the product to be considered damaged. A Vertical Shock Test System can be used to test fragility according to ASTM D3332.

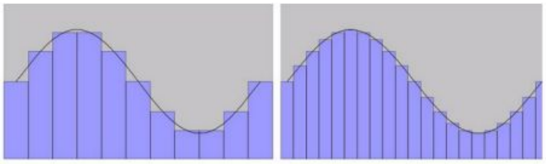
4.1.3 Finite Element Method

In order to test numerous packaging and product designs, without spending the money on manufacturing, the team conducted a simulation in SolidWorks using Finite Element Method. Having no previous experience or training with Finite Element Analysis the team needed to learn what it was, how it worked and how drop testing could be simulated within SolidWorks. To develop our understanding of this software, Cisco provided us with an informative lecture, <u>Introduction to Finite Element Analysis & Tools</u>. FEA is a useful simulation tool because it simplifies complicated parts into small finite elements that can be processed by a computer.

Using matrix equations to solve for stress, strain and displacement within these finite elements, FEA constructs an approximation of reality.

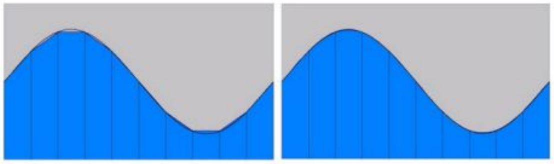
The key to conducting a FEA experiment is a valid base model. The outputs of a smultation are only as good as its input. Thus, the proper parameters needed to be defined. Each part within the SolidWorks Assembly needed to be broken into simple shapes (meshed). All parts needed to be constrained to represent the physical product and the supporting packaging needed to be defined. Upon the proper design of the previous three parameters the base model was validated with a comparison between simulation results and hand calculations. The base model of the current design was used as a baseline to compare the results of new designs to the current design. While the accuracy of the individual model is subject to question, with precision and consistency in the modeling of new designs comparisons could be drawn between the design alternatives.

To accurately model a drop test in FEA we need to define the mesh of all parts with an element type and size. Element types in our model are either a shell or a solid tetrahedral. Shells are a simplified solid part that represent the shell of the part. A mesh determines the number of elements in each part. As the mesh is refined the model becomes more and more accurate. There are two methods of refinement. H refinement uses many simple elements. To refine the mesh these simple elements are divided in half to better approximate the solution. P refinement uses few complex elements and improves output accuracy by increasing the complexity of each element. Figure 14 provided a graphic on the difference between a coarse mesh vs. a fine mesh for the two methods of refinement. We refined our model using the H method.



H-Method with Course Mesh

H-Method with Fine Mesh



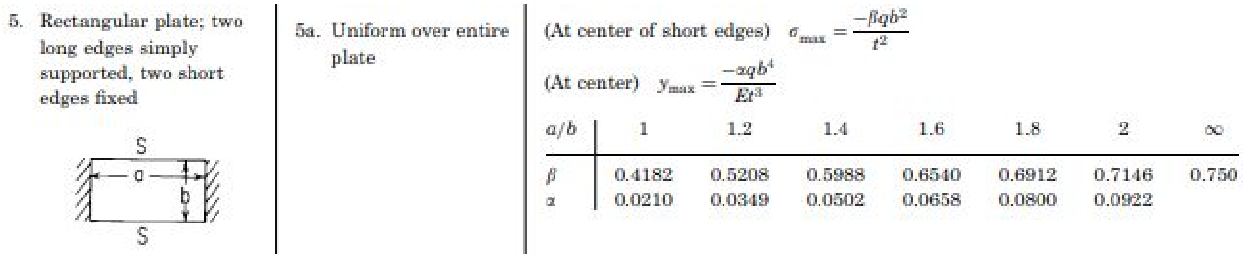
P-Method with 2nd Order Polynomial

P-Method with 3rd Order Polynomial

Figure 14: Coarse vs. Fine Mesh for H & P Refinement

4.1.4 Model Validation

The biggest take away from our accelerated FEA course was that our base model needed to be valid, or represent the real world. One validation step was measuring if the FEA results were within 20-30% of the Roark's Formula hand calculations. Using Roark's Formulas for Stress and Strain (Figure *15*), we compared the maximum stress along the front edge of the chassis between our hand calculation and our FEA results. Upon finding that these numbers were within 20-30% error we were one step closer to validating our base model.



5. Rectangular plate; two long edges simply supported, two short edges fixed

5a. Uniform over entire plate

(At center of short edges) $\sigma_{\max} = \frac{-\beta q b^2}{t^2}$

(At center) $y_{\max} = \frac{-\alpha q b^4}{E t^3}$

a/b	1	1.2	1.4	1.6	1.8	2	∞
β	0.4182	0.5208	0.5988	0.6540	0.6912	0.7146	0.750
α	0.0210	0.0349	0.0502	0.0658	0.0800	0.0922	

Figure 15: Hand Calculation Method Used & Maximum Stress Formula

4.2 Expert Knowledge

Three Cal Poly professors and Henry Lee of Cisco guided our solution design, FEA design, constraints and validation throughout this project.

4.2.1 Drop Test Design

Dr. Peter Schuster, a mechanical engineering professor at California Polytechnic State University, San Luis Obispo, guided the initial design of our FEA simulation on the SolidWorks model. In our attempt to simulate a drop test he proposed we replicate the Top Up and Bottom Up drop, similar to Cisco's packaging studies.

4.2.2 Brainstorm of Solution Design

Professor Georgeau also aided us in the brainstorming of solution designs. He informed us on potential features that could be added to the manufacturing process such as beading and ribbing. Beading is a raise or sink in a portion of sheet metal that is manufactured with stamping or dies. Its purpose is to increase rigidity in the sheet metal cover of the chassis. Ribbing is used internally and is placed on the corners of the chassis to provide structural support and prevent bending.

4.2.3 FEA Design & Validation

Dr. Peter Schuster a Cal Poly Mechanical Engineering Professor, Dr. Garrett Hall a Cal Poly Civil Engineering Professor and Henry Lee an Engineer at Cisco aided our group in FEA test design, and validation.

Dr. Garrett Hall guided our team through design. Initially, our group was focused on conducting a drop test within FEA. Being that we would need to simulate a dynamic environment our simulation time spanned 80 days for one model. Dr. Hall recommended our group use a linear static model. A linear static model kept the product fixed in time. Instead simulating dropping the package, we applied a gravitational force to the model to approximate the force the package experienced during a drop test. This method would be applied to both the Top Up and Bottom Up drops. The acceleration applied was 30 G's.

Dr. Peter Schuster aided our team in simplification of our model and properly fixing the model. The product has many parts that contribute to its function, but add little structural support. By removing numerous parts that were not required for the simulations function and representing them with forces our simulation time was reduced. The simplified model contained the top and bottom covers, the main PCB, and the sheet metal components on the backside of the chassis; there were a total of 11 parts modeled. Also, Dr. Schuster provided insight as to how to simulate bolts and rivets on the chassis.

Henry Lee was very helpful in providing information to the team. Mr. Lee provided insight on simplifying our model’s mesh and validation of our results. By simplifying the mesh of the sheet metal cover that the force was being applied to a shell instead of a solid, we simplified our model. Also, Henry Lee provided a method to validate our results. He recommended using Roark’s Formulas for stress and strain to compare hand calculations results for stress to results found in our model. If these numbers were within 30 percent, our model was considered valid.

5. Solution Design

Our goal was to create comprehensive solutions that did not affect the functionality of the unit and eliminated all failures. The design criteria that Cisco provided allowed us to narrow our focus by ruling out designs that would not be feasible. Examples of these included designs which changed the overall dimensions of the unit, the mounting style of the top cover, the clearance from the internal components, and/or involved packaging changes. Thus, we decided to focus our attention on strengthening the top and bottom covers since they create the main structure of the product. Additionally, we decided to produce two different solutions: a short term solution which could immediately address the problem in a matter of weeks and a long term solution that was more cost effective, but would require longer to implement.

5.1 Design Constraints

We faced several different constraints that we needed to adhere to when developing our solutions. First, overall dimensions could not be modified since this product comes in an industry standard size of 1RU (1 Rack Unit) to ensure fit into any server rack the customer buys. The maximum dimensions of a 1RU sized chassis are approximately 1.75" height x 17.5" wide with varying depths. Additionally, the screw pattern used to attach the top cover was specifically designed to prevent EMI (electromagnetic interference) from other components, which could stop the unit from turning on. Changing how the top cover is attached would require knowledge of electrical engineering that is beyond our scope. Furthermore, all of the electrical components in the unit require clearance from the metal chassis to prevent short circuiting and to protect against impacts, so clearances could not be modified. Finally, strategically modifying the packaging would require a deep understanding of packaging engineering, so we decided to leave it as is.

5.2 Design Process

We conducted two rounds of design and simulation. In both rounds, our objective was to increase and improve the rigidity of the unit as well as decrease the stress it experiences during shipping. Using expert advice and knowledge gained from our literature review, our first design round focused on modifying and/or adding various features to the top cover as well as developing an external brace to support the chassis. After running simulations on the first round designs, the best performing designs were kept and tweaked based on their performance. The way testing and recording data was improved for the second round of testing.

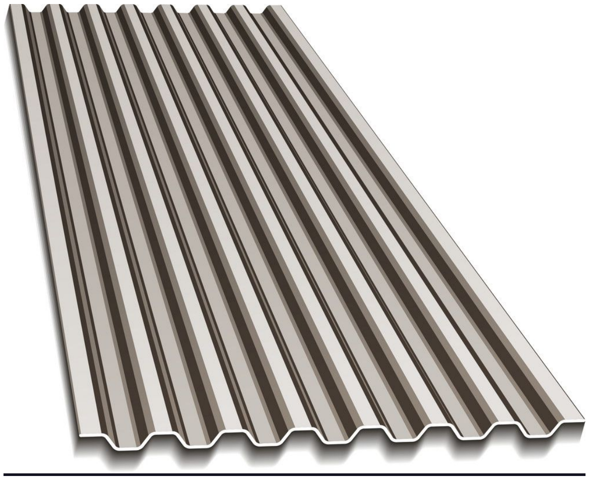


Figure 16: Corrugated Metal

When developing the second round of designs, Professor Georgeou suggested beading as a low cost feature that would add strength and rigidity. In the same way the shape of corrugated metal is the source of its strength, beading strengthens surrounding areas by leaving indentations that can prevent bending. This wave shape is only strong when bending perpendicular to the cross section of the wave shape, direction and placement of the beads have a large effect on their ability to resist bending. Designs were created with beads running laterally as well as some running front to back, in varying numbers, to understand how much strength they added. Implementing these changes would require expensive and time consuming changes to the manufacturing, so we developed a solution that could be implemented relatively quickly and did not require changes to the product, an external brace.

5.3 Designs Considered

The final designs under consideration were the best performers from the first round of designs that had been tweaked to improve results. Many of these designs utilize beads to add strength to the product, one employs a brace to take some of the load off of the chassis.

5.3.1 Lateral Beads

Designs with beads running across the width of the top aimed to redistribute forces to the sides of the chassis, as the sides are strong and already low stress, reducing the load in high stress areas like the front supports. The first of these designs placed 2 beads straddling airflow perforations in the front of the top cover, stopping 1/2" from both sides of the top. The following designs had an additional 2, 4, and 6 beads placed directly behind the front 2, with the goal of understanding the ideal number of beads that reduced stress to manageable levels.

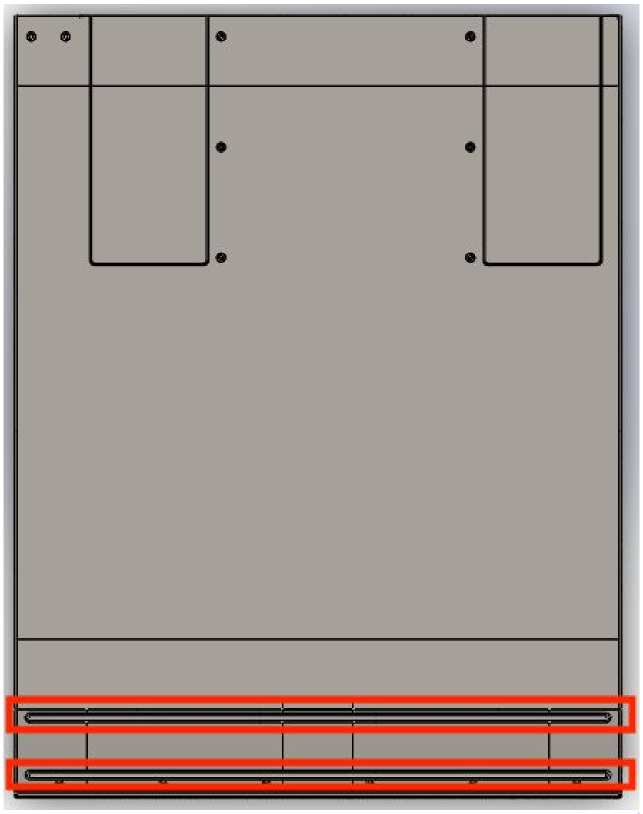


Figure 17: Design with 2 Lateral Beads, Top Down View

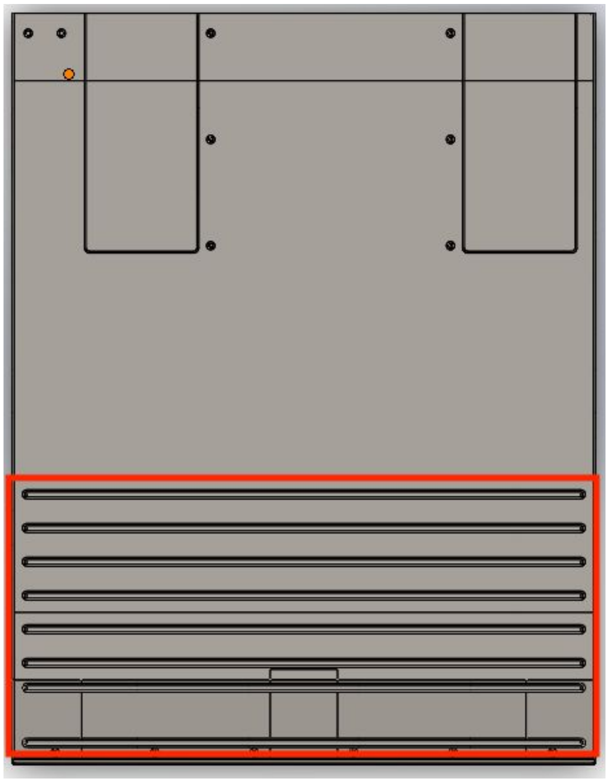


Figure 18: Design with 8 Lateral Beads, Top Down View

5.3.2 Front to Back Beads

Designs with beads running front to back were created, even though they would redistribute load to the weak front and strong back of the chassis, purely to understand their effect. These beads stopped 1/2" from the power supplies in the back of the chassis, which fill the entire internal vertical space, and 1/2" from an angled section at the front, which slopes down to meet the front supports. These designs also had 2, 4, 6, or 8 beads evenly space across the width of the top cover.

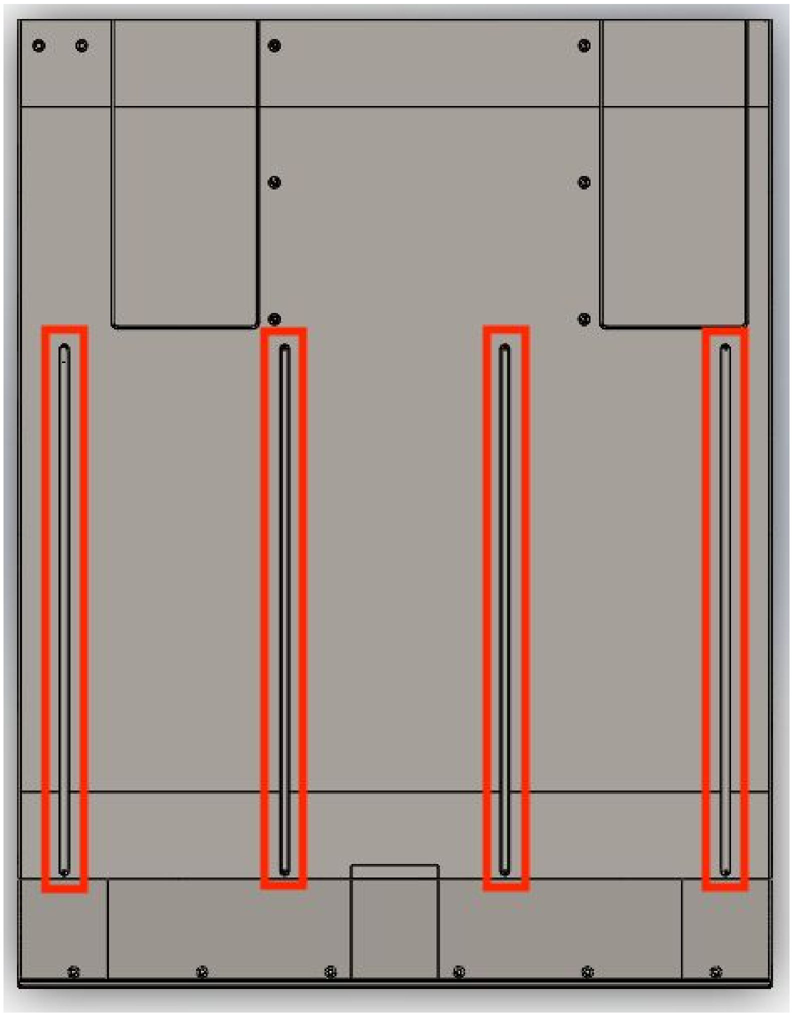


Figure 19: Design with 4 Front to Back Beads, Top Down View

5.3.3 External Brace

A design utilizing an external brace was created, using existing, reinforced screw holes on the sides to grip the chassis. The brace was made from the same sheet metal as the rest of the chassis, sported two lateral beads to add strength and rigidity, and would be installed in the front set of mounting screw holes. The brace sat on top of the chassis with .2" clearnance so that it would be in tension if the failure mode Cisco identified occurred. If it were underneath the chassis, it would experience compression and would risk buckling, reducing effectiveness. The customer would remove and recycle the brace before installation.

Figure 20: External Brace Fitted to Nexus 9200

5.3.4 Redistributed Front Supports

Designs were created that redistributed the front supports so they were less clustered and shared the load more evenly. This included making them all of equal size, as the existing design included 3 thicker supports clustered within 4" of each other on the right and 2 thinner supports spread across the remaining ~13". The front facing ports would need to be moved to accommodate this and would result in components of similar function not being next to each other, which customers may not like. While this goes against Cisco's criteria of moving internal components, we felt it had enough potential to devote our time.

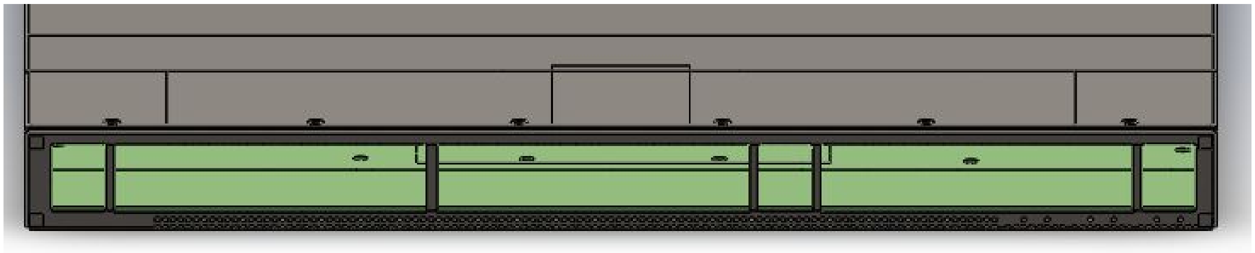


Figure 21: Design with Rearranged Front Supports

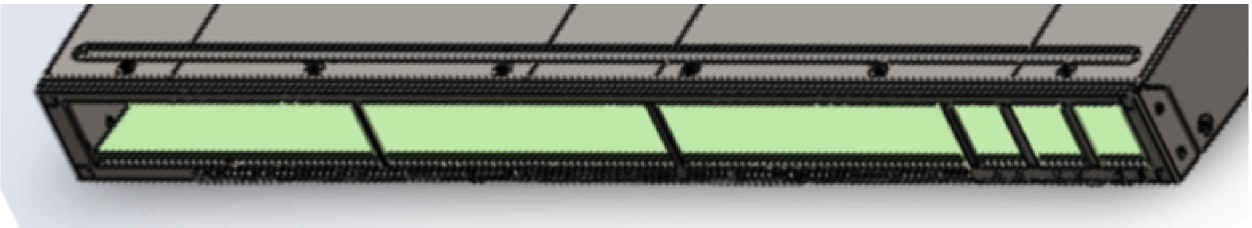


Figure 22: Current Design with Unchanged Front Supports

5.3.5 Thicker Top

A design with a 0.0456" thick top cover was creating which aimed at preventing deformation by strong arming it. Besides the blunt force idea of making it thicker, we aimed to reduce stress in the critical areas by strengthening the faces pressing on them, like the top, causing these stresses. While this solution is not applicable to the Nexus 9200 product line, testing how much thickness effects strength and stress would be very helpful in crafting design guidelines for future products.

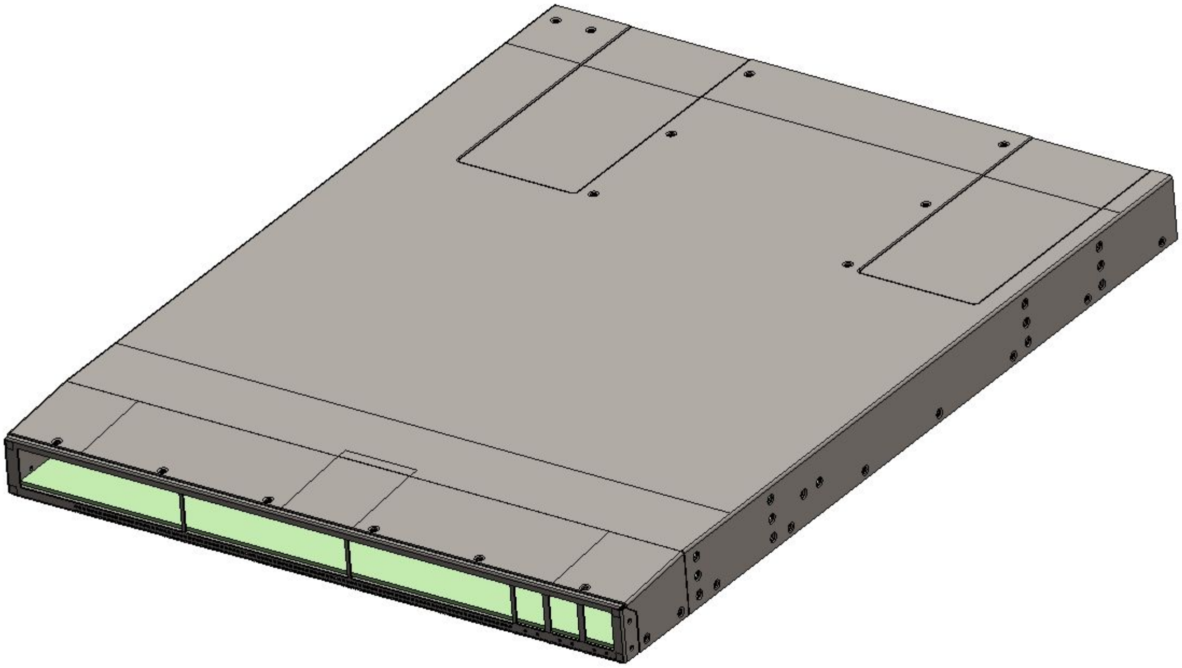


Figure 23: Design with 0.0456’’ Thick Top Cover

5.3.6 New Foam Contact Pattern

A design with a new foam contact pattern was created with the goal of supporting the unit in the area it deformed the most. This area was the center line running from front to back. Modifying the current foam packaging design, we rotated the foam 90° so there was more foam closer to this center line on the top and the bottom of the unit. While the new supports are longer, they were made thinner, to ensure the same surface area of contact as the current design. This was done because having too much foam relative to the weight and size of the unit could result in a higher deceleration, impulse, and forces experienced when dropped, which would increase the stresses experienced.

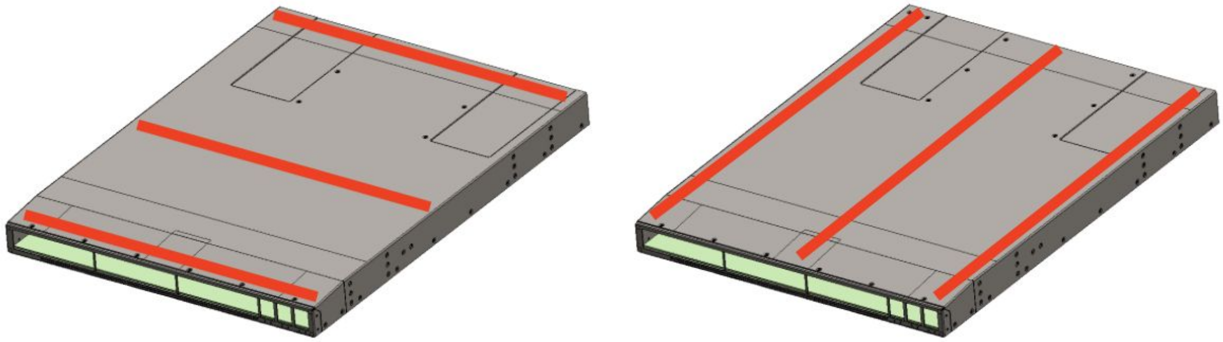


Figure 24: Current Bottom Foam Supports vs New Bottom Foam Supports

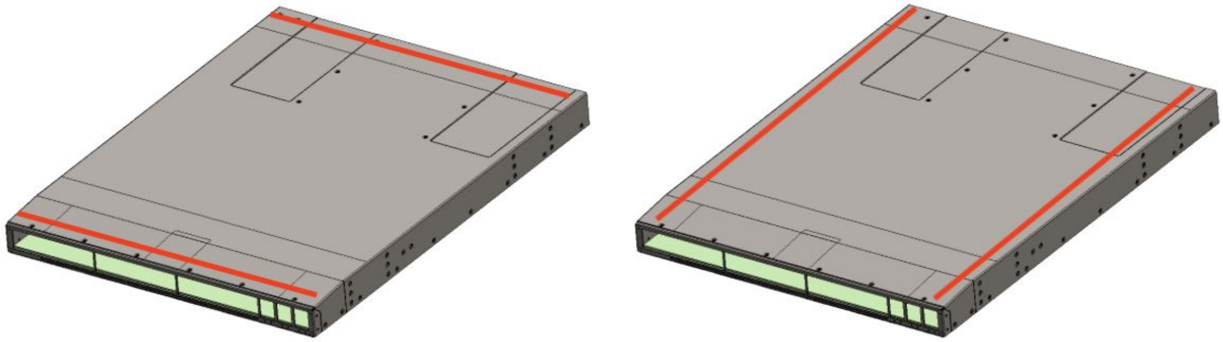


Figure 25: Current Top Foam Supports vs New Top Foam Supports

6. Test and Evaluation of Design Alternatives

In order to create a test to evaluate the different design ideas, a computer model was created to simulate the real life drop. After creating the model in the computer it had to be verified with several methods in order to make sure that it was indeed accurate. After the model was deemed accurate, testing criteria and an evaluation methodology was decided upon. Additional tests outside of the alternative designs were conducted to understand the model better.

6.1 FEA Model

In order to simulate the drop in the computer, the Finite Element Method was chosen. In order to create the FEA model the software, simulation type, test setup, and model had to be defined and designed.

6.1.1 Software

The first step to creating and testing a model was selecting software packages to design a three dimensional model and to simulate a dropping chassis with the Finite Element Method. For this project, SolidWorks was used for both the design and analysis.

Solidworks is an industry standard for three dimensional design and is both extremely versatile and easy to use; our team also had considerable experience with the software.

Solidworks is not an industry standard for simulation software. However, our team had no experience with simulation software and Solidworks' simple user interface made it the clear choice for this project. It should be noted that Solidworks has some missing features that affected the design and analysis of our experiment; these will be commented on later.

6.1.2 Simulation Type

The Finite Element Method is a simulation method that breaks complex shapes into smaller, simpler ones and uses matrix computations to solve for stress, strain, and displacements. For these testing procedures, a linear static simulation method was chosen.

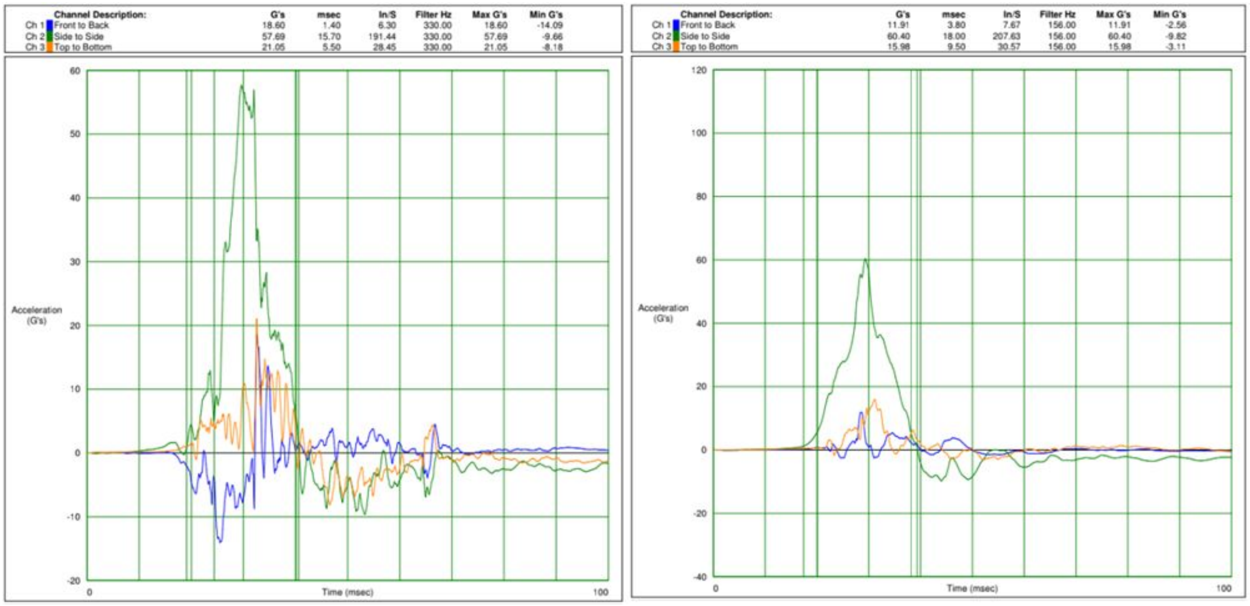
Linear means that the relation between stress and strain for all materials in the simulation is assumed to be linear. Linear approximations reduce computing times and are appropriate when tests are assumed to be operating in the elastic region of the their respective stress strain curves. For this model the foam of the packaging violates this assumption and further steps were taken in the validation of our model to address these issues.

A static simulation means that it does not take place over a time interval. Again, this vastly simplifies processing times and is most valid when materials being test have a low strain rate reliance such as the steel in the model. The foam in this model violates this assumption and further steps were taken in the validation of our model to address these issues.

6.1.3 Test Setup

Linear Static simulations in Solidworks require that elements are fixed in space and thus a drop could not be exactly replicated. Thus an approximation of the drop needed to be modeled and the orientations needed to be narrowed down.

Drop test data for the N9K switch from Cisco System was examined to determine the severity of each of the orientations. The maximum acceleration was seen in the top-side up and top-side down drops at about 60 G’s. Figure 26 shows the drop test data acquired from Cisco Systems. These two worse case scenario drops were chosen to be modeled in our FEA software.



Channel Description:	G's	msec	In/S	Filter Hz	Max G's	Min G's
Ch 1 🟦 Front to Back	18.60	1.40	6.30	330.00	18.60	-14.09
Ch 2 🟩 Side to Side	57.69	15.70	191.44	330.00	57.69	-9.66
Ch 3 🟧 Top to Bottom	21.05	5.50	28.45	330.00	21.05	-8.18

Channel Description:	G's	msec	In/S	Filter Hz	Max G's	Min G's
Ch 1 🟦 Front to Back	11.91	3.80	7.67	156.00	11.91	-2.56
Ch 2 🟩 Side to Side	60.40	18.00	207.63	156.00	60.40	-9.82
Ch 3 🟧 Top to Bottom	15.98	9.50	30.57	156.00	15.98	-3.11

Figure 26: Drop Test Data of Top Up and Top Down Drop (Left to Right)

In the figure, from left to right, shows the top-up drop and top-down drop for an N9K switch weighing 29.5 lbs. Both drops resemble a half sine curve, last about 20 ms, and peaks at about 60 G’s.

In order to replicate a drop in static simulation, the unit was modeled on top of foam fixtures with a large gravitational force. The sensor for the drop data collected by Cisco was placed inside of the unit of the chassis and not the package itself. Assuming that the foam the chassis sits on is perfectly elastic, then force of gravity on the unit is half of the maximum acceleration measured - 30G’s. Two drops were modeled - a top-up drop and a top-down drop for each alternative.

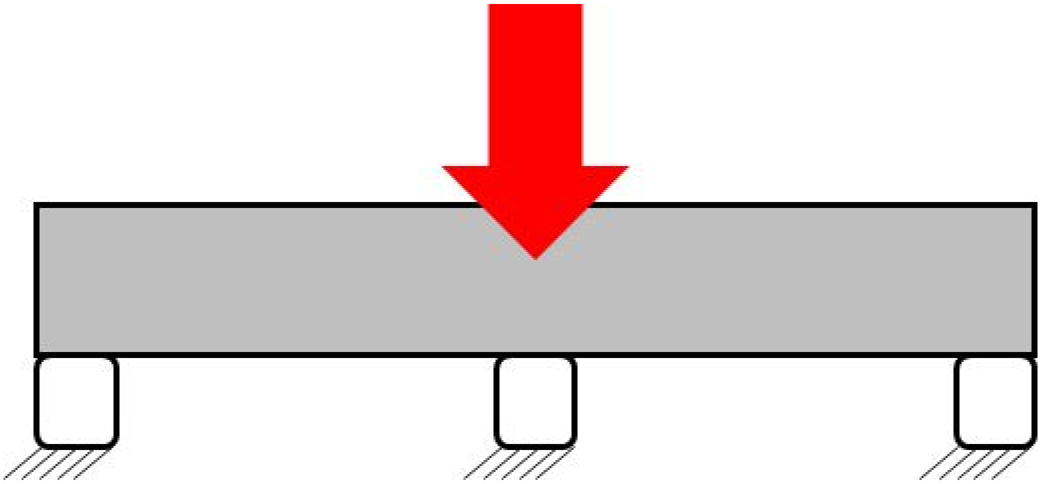


Figure 27: Free Body Diagram of Static Approximation

Figure 27 shows the free body diagram for the linear static simulation. The red arrow represents the force applied by gravity. Essentially, gravity is pressing the chassis into foam supports, which have an opposite and equal normal reaction.

6.1.4 3D Model

Before simulating a drop, the chassis had to be remodeled and simplified so that it could be easily modified for different design alternatives and to minimize the computing time. All sheet metals components were modeled along with the main circuit board. Other circuit boards, heat sinks, power supplies, ports, bolts, rivets, and fans were not modeled and were replaced with equivalent forces in the model. Tiny features including small filets and holes were not modeled. Generally speaking the model is more accurate towards the front of the chassis where the deformations and damage was happening in real life. Figure 28 shows the components that were modeled for testing in Solidworks compared with the actual chassis.

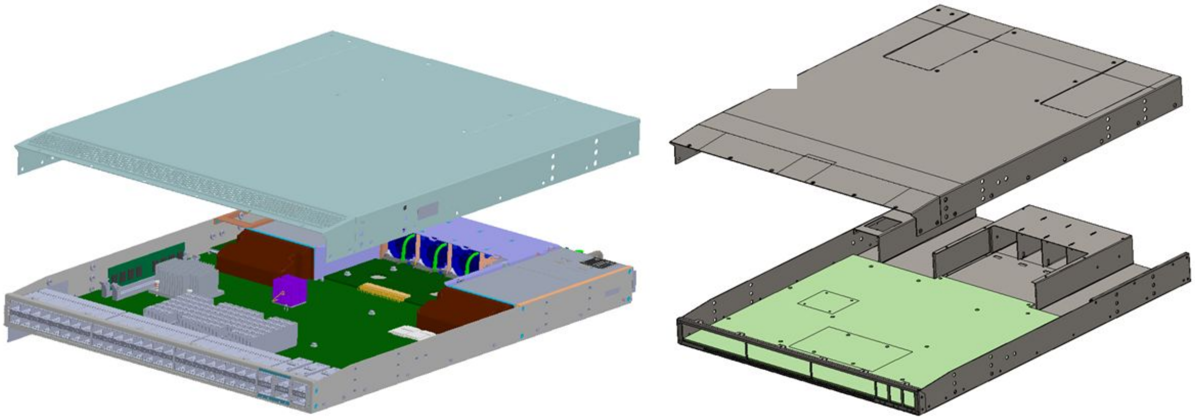


Figure 28: Original Vs. Simplified Model (Left to Right)

6.1.5 Mesh

Various mesh controls were applied to create an accurate and fast to compute model. There are many different types of meshes, but Solidworks only has three - a solid tetrahedral, two dimensional triangles, and beam elements.

Meshes in contact with fixtures can not be two dimensional in Solidworks. This makes modeling sheet metal in contact with foam difficult. The preferred method for modeling a thin sheet with solid elements is with brick elements an even number of layers thick and greater or equal to four layers. However, because Solidworks does not allow the use of brick elements, tetrahedral ones were used instead at two layers thick. This means that there is no neutral axis in the parts modeled this way. This methodology tended to vastly underestimate the stresses in the model.

The top and bottom pieces were modeled as two dimensional curvature based mesh or a solid tetrahedral mesh depending on if the drop test was a top-up drop or top-down drop. Curvature based mesh means that the size of the elements are reduced on complicated surfaces. The mesh in contact with the fixtures (foam) were modeled as solid elements. Since this creates two fundamentally different models, one which has a solid meshed top cover and one that has a solid meshed bottom cover, the top-up and top-down drop tests can not be directly compared against each other. Additionally areas on the top cover can not be compared against areas on the bottom plate.

An aspect ratio plot of the mesh was constructed and evaluated. The aspect ratio of a tetrahedral is the ratio between the largest and shortest distance between any vertex and base. Tetrahedrals and triangles that are uniform in size will have an aspect ratio close to one, while distorted tetrahedrals and triangles will have ratios higher than four. A model has a good aspect ratio if its elements do not exceed four. Figure 29, from left to right, shows the aspect ratio for the top-down and top-up drop tests on the base model. The plot is a rainbow heatmap from 0 to 10. No values are above 4.

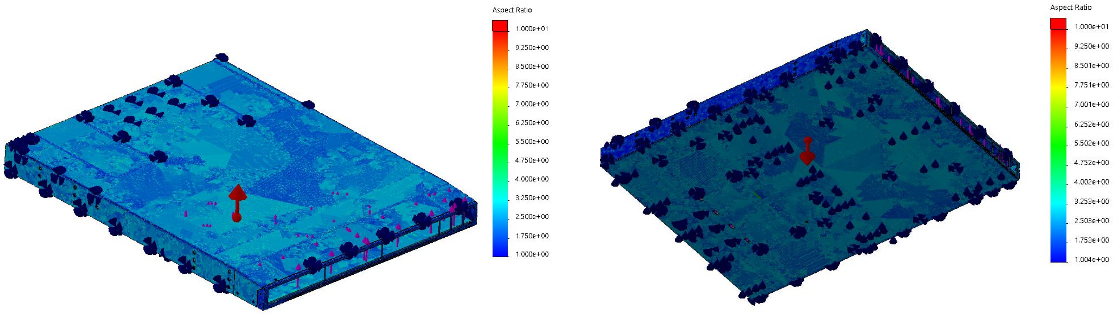


Figure 29:Aspect Ratio Heat Map Top and Bottom View

Figure 30 shows the aspect ratio of the inner components of the base model. The components in the back are a coarser mesh and have a high aspect ratio. However, the areas of interest are far away from these points and their accuracy does not affect the accuracy of the areas of interest.

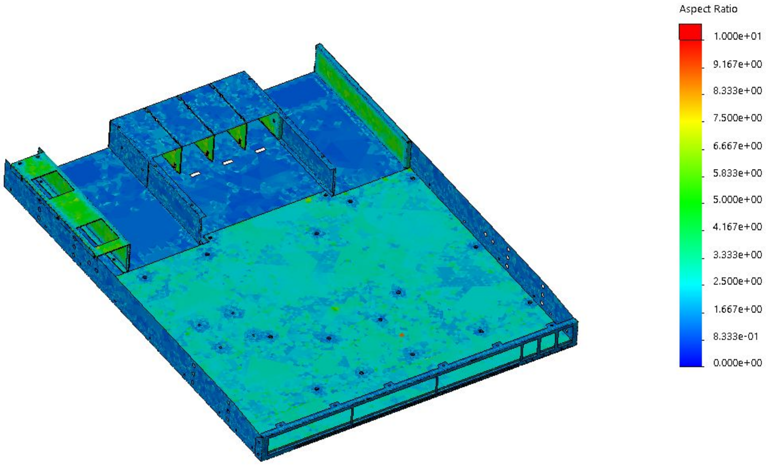


Figure 30: Aspect Ratio Heat Map Inside View

6.1.6 Connectors

Rivets and screws were simulated using rigid connectors. This means that selected pairs of cylindrical cutouts in the sheet metal were constrained with respect to each other, but not with the rest of the model. These connectors are not an accurate representation of real life connectors such as screws and rivets used in the chassis and thus the areas directly surrounding the holes have exaggerated stresses; results become more accurate at a distance of about one and a half times the radius of the hole being constrained. For this project's purposes, this is the preferred way of modeling connectors and has little impact on the accuracy of the results.

6.1.7 Contacts

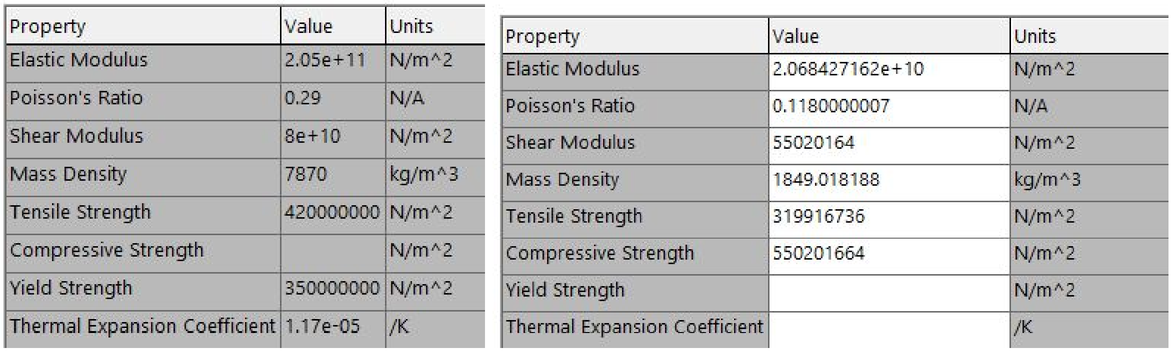
There are no simulated contacts between plates in the model. Model parts are allowed to pass through each other in the simulation. Simulating no penetration contacts greatly increases the simulation time without affecting results very much and thus was not chosen for this model.

6.1.8 Fixtures

In this simulation, foam was replaced with elastic fixtures. Fixtures constrain the model in three dimensional space and make the matrix equations solvable for the computer. Elastic fixtures were applied where the foam touches the chassis in real life and had a stiffness of 10.66 lb/in^2 and a shear stiffness of 1lb/in^2. As mentioned earlier foam does not fit well into a linear static simulation. Because the foam had approximated material properties in the simulation and did not fit well into the model, additional model validation was needed to justify model results.

6.1.9 Material Properties

In order to create a model for the simulation, the material properties for the chassis must be emulated. All of the sheet metal parts in the assembly are made of AISI 1020 cold rolled steel. Table 7, on the left, shows the material properties for the steel used in the model. The other material simulated was the circuit board which was based off of FR4 circuit board. Table 7, on the right, shows the material properties for the main circuit board that was simulated. In a linear static model only Young's modulus, and poisson's ratio are used.



Property	Value	Units
Elastic Modulus	2.05e+11	N/m^2
Poisson's Ratio	0.29	N/A
Shear Modulus	8e+10	N/m^2
Mass Density	7870	kg/m^3
Tensile Strength	420000000	N/m^2
Compressive Strength		N/m^2
Yield Strength	350000000	N/m^2
Thermal Expansion Coefficient	1.17e-05	/K

Property	Value	Units
Elastic Modulus	2.068427162e+10	N/m^2
Poisson's Ratio	0.1180000007	N/A
Shear Modulus	55020164	N/m^2
Mass Density	1849.018188	kg/m^3
Tensile Strength	319916736	N/m^2
Compressive Strength	550201664	N/m^2
Yield Strength		N/m^2
Thermal Expansion Coefficient		/K

Table 7: Material Properties For Sheet Metal and PCB

6.2 Model Validation

Once an FEA model was constructed in the computer, it was evaluated and tested to see if it was an accurate approximation of a real world drop. Three types of validation methods were used to verify the accuracy of the model; hand calculations, solution boundaries, and convergence plots were used to validate the model.

6.2.1 Roark's Formulas

Utilizing Roark's Formulas for Stress and Strain we calculated the max stress within the chassis using the equation for a rectangular plate fixed at two short edges. The maximum stress is calculated at the center point of the short edges. This is where we will probe the chassis to gather the simulation results. Our goal is for our hand calculated value, accepted value, and simulated value, experimental value, to be with 30% error of each other.

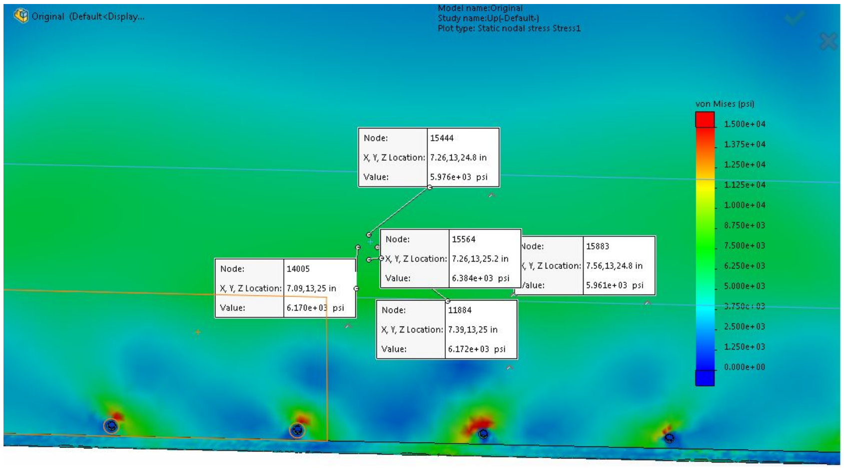
The parameters of this equation are defined as follows:

σ_{max} = maximum stress
β = ratio of sheet metal length (a) to sheet metal width (b)
q = distributed force applied to the sheet metal
a = sheet metal length
b = sheet metal width
t = sheet metal thickness

The calculated values for the base model were:
σ_{max} = **32302276.24** N/m^2 **= 4685.057506 psi**
β = **0.56907**
q = 30*(9.8 $meters/second^2$) = **294 Newtons**
a = **0.56079 meters**

b = **0.43940 meters**
t = **0.00100 meters**

The calculated or accepted max stress in the center of a short side of the chassis is **4685.057506 psi.** Our measured or experimental value of max stress from the top upright drop was **6132.6 psi.** (*Figure 27*) Using the percent error equation our simulation resulted in a **30.9% error.** Our target was to be within 30% error, however being only 0.9% over our goal, we accepted the result and validated our base model.


Model name:Original
Study name:Up(-Default-)
Plot type: Static nodal stress Stress1

Node: 15444
X, Y, Z Location: 7.26,13,24.8 in
Value: 5.976e+03 psi

Node: 15564
X, Y, Z Location: 7.26,13,25.2 in
Value: 6.384e+03 psi

Node: 15883
X, Y, Z Location: 7.56,13,24.8 in
Value: 5.961e+03 psi

Node: 14005
X, Y, Z Location: 7.09,13,25 in
Value: 6.170e+03 psi

Node: 11884
X, Y, Z Location: 7.39,13,25 in
Value: 6.172e+03 psi

Figure 31: Results from Base Model Up - Location 2

6.2.2 Solution Boundaries

As mentioned before, the foam in the simulation provides a unique problem. It is strain rate reliant, experiences large deformations, and does not have a linear stress strain relationship in the elastic region, and is thus not suitable for a linear static study.

In order to understand the variance in the study caused by the foam, two extreme approximations for the foam were made; one in which its stress strain curve was nearly perfectly elastic and the other where it was perfectly inelastic. For each measured area on the model, the true stress of the foam would be somewhere in between these extremes.

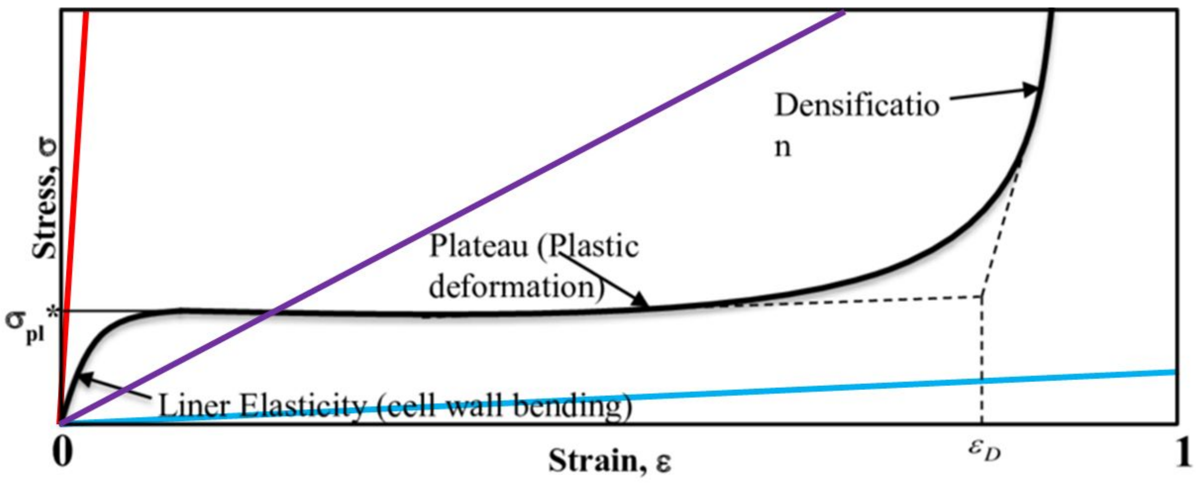


Figure 32: Stress Strain Curve Of Foam Compared with Other Approximations

Figure 32 shows the general shape of a stress strain curve for a foam material. The red line represents the foam being modeled as a perfectly inelastic fixture and the blue line represents a nearly perfectly elastic fixture to simulate the foam. The purple line is the simulated elastic fixture in the model. Knowing that the true stress strain relationship lies somewhere between the red and blue extreme approximations, the variance and boundaries for the modeled foam can be approximated.

6.3 Comparison of Alternatives

Once the alternative designs were constructed, a standard way to test and evaluate them was needed. It was decided after various meetings with the stakeholders that the criteria that would be evaluated for each model was robustness, cost, sustainability and implementation time. Long term and short term solutions would weight these criteria differently.

6.3.1 Robustness

The first step in comparing robustness between models was to first evaluate the areas of high stress on the control test. After two rounds of testing, twelve areas of high stress were chosen near the front of the chassis. The areas of high stress were consistent across the top-up drop and top-down drop. Figure 33 shows the chosen areas on the top and bottom of the front of the chassis.

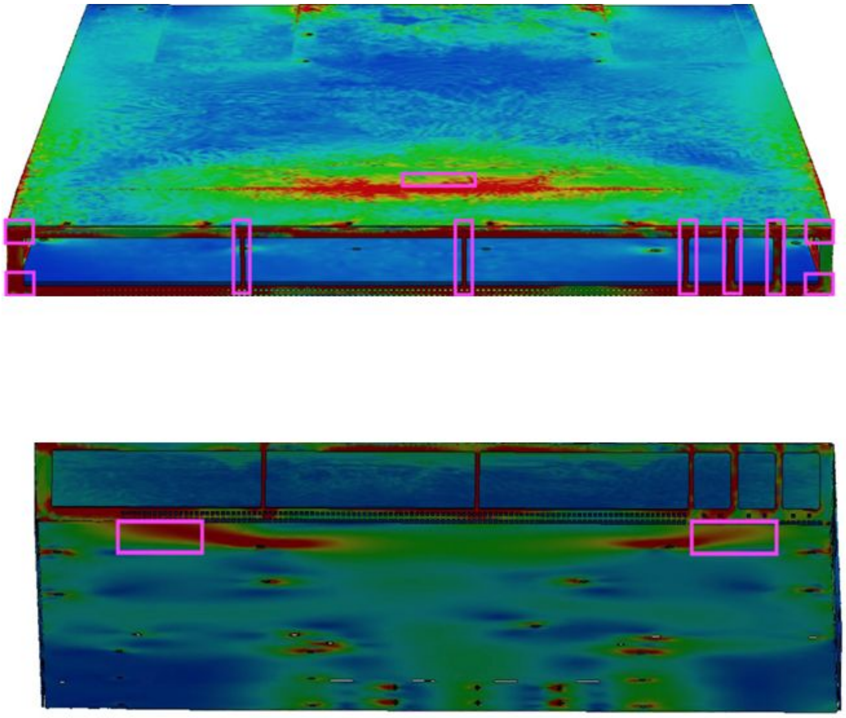


Figure 33: Sites of High Stress on Base Model

As mentioned before areas of interest can not be compared between the top and bottom cover and between the top-up and top-drown drop. This means that there are four distinct categories that can not be directly compared, but need to be factored in for robustness.

Figure 34 shows the control model top-up drop and control model top-down drop from left to right. The “Top Front Middle” bar is colored a different color to show that it can not be compared to other points of interest on the chassis. From this data we can conclude that the front supports are the most critical areas in both types of drops. Across all tests, these results were true.

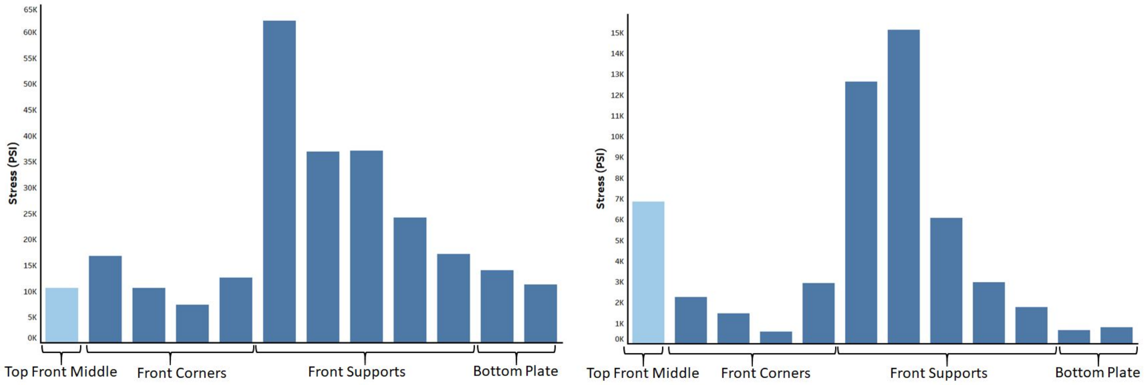
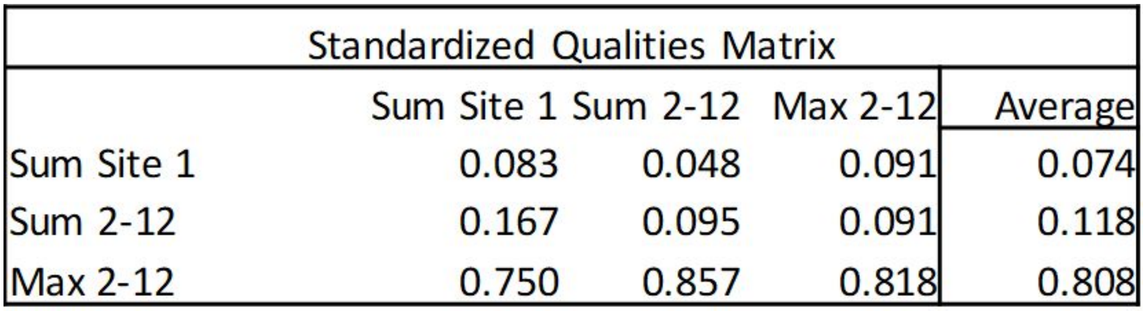


Figure 34: Top Up and Top Down Drops of Base Model (Left to Right)

Site 1 refers to the top middle of the chassis and sites 2-12 refer to the other test sites on the bottom cover. In order to assess the robustness of each design, the stress of site 1, the sum of the stress across sites 2-12, and the maximum value between sites 2-12 across both sets of drops were weighted and ranked. Top-up and Top-down drops had equal weighting and their values were summed. 2-12.

An analytical hierarchy process was then used to determine the most robust alternatives. Table 8 shows the standardized values of the qualities of each model used to evaluate its robustness.

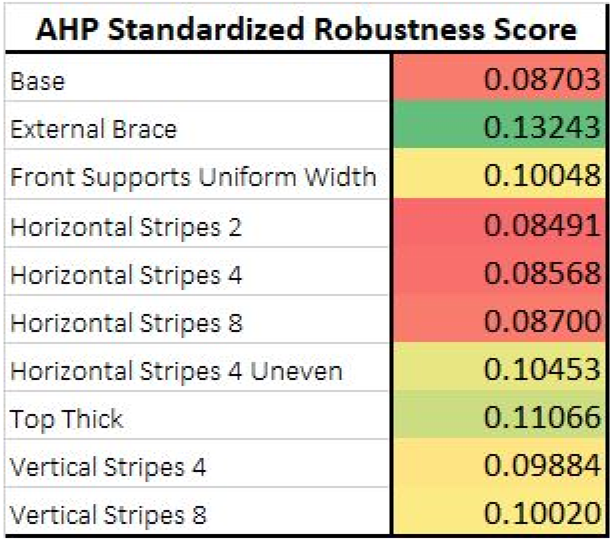


Standardized Qualities Matrix				
	Sum Site 1	Sum 2-12	Max 2-12	Average
Sum Site 1	0.083	0.048	0.091	0.074
Sum 2-12	0.167	0.095	0.091	0.118
Max 2-12	0.750	0.857	0.818	0.808

Table 8: Average Standardized Qualities for AHP of Robustness

A reduction in the maximum stress value was given the highest weight in the model, as it is the main measure of mechanical failure. The sum of sites 2-12 were considered slightly more important that site 1 as there were more data points that went into it.

Each model was the ranked and standardized based on each quality listed above. The standardized averages for each solution alternative and the standardized values for the qualities were combined to give a standardized robustness score for each solution alternative. Table 9 shows the standardized values found in the AHP process for each solution alternative.



AHP Standardized Robustness Score	
Base	0.08703
External Brace	0.13243
Front Supports Uniform Width	0.10048
Horizontal Stripes 2	0.08491
Horizontal Stripes 4	0.08568
Horizontal Stripes 8	0.08700
Horizontal Stripes 4 Uneven	0.10453
Top Thick	0.11066
Vertical Stripes 4	0.09884
Vertical Stripes 8	0.10020

Table 9: Standardized Robustness Score for Each Solution Alternative

From here the order of robustness of each design aspect is clear. It should be noted that more than one alternative can be selected, but the interactions between them is unknown. It should also be noted that some alternatives conflict; putting horizontal stripes on the chassis would not allow vertical stripes.

6.3.2 Economics

Another crucial metric that was used when evaluating different alternatives was the cost of implementing each solution. First and foremost, if the cost of implementing a solution was greater than the per unit cost of the problem, it made no sense to pursue the solution and was immediately discarded. Furthermore, with limited access to manufacturing data, we made two crucial assumptions when calculating and comparing the costs of alternative solutions. The first assumption was that the cost of implementing bead rolling of any kind (i.e. "Horizontal Stripes 2," "Vertical Stripes 8," etc.) was $0 per unit. This assumption was made because it requires minor changes to existing dies. Given the volume of yearly unit production (~XXX,XXX per year), the one time equipment modification fee is effectively zero. Additionally, we also assumed the cost of evenly spacing the front supports to be $0 per unit. Similarly, satisfying engineering change orders to redesign the placement of the front supports is a one time labor and equipment cost that will effectively zero out on a per unit basis over time. Finally, in calculating costs of implementing the remaining solutions (the external brace and changing the thickness of the sheet metal), we used cost data that was provided to us to estimate the per gram cost of sheet metal that Cisco would pay. We then multiplied that value by the weight of the two remaining solutions. A breakdown of those costs will be shown is shown in Table 14 and Table 15 respectively.

6.3.3 Implementation Time

We were able to estimate time to implement changes with the professors and experts that helped us by considering the size, shape, complexity, and changes to the product. Time to implement increases as any of those factors increase. We estimated that it would take 2 months to implement the external brace, whether it was manufactured by Cisco's contracted factories or by a 3rd party, since it does not require engineering change orders and is a relatively simple part to manufacture. Beads are a significant step up as they require minor product changes, but need redesigned dies. This pushes implementation time for beading to around 3 months. Increasing the sheet metal thickness of the top requires significant changes to the product, stamping dies, reengineering of the chassis, and packaging drop testing. We estimated implementation for this would take around 4 months. If time allowed, we would have gotten estimates from these vendors and used that to compare the potential solutions.

Because the estimates for implementation time are rough, the implementation time will be compared against each alternative solution in an analytical hierarchy process. Instead of classifying an external brace to take 2 months, the implementation time of the brace can be rated to be significantly faster than an added bead to the product.

6.3.4 Sustainability

As previously mentioned, the final criteria we used to compare alternative solutions was sustainability. The question we wanted to address was “How sustainable is this solution over a long period of time?” Solutions that we felt were not very sustainable (i.e. environmental reasons, cost, etc.) were either discarded or used as a short term solution. On the contrary, modifications that we felt were sustainable were considered as a long term solution. The external brace and the thicker top sheet metal were the only alternatives that had an impact on sustainability. The external brace requires an entirely new metal part to be manufactured and would only be used during shipping. There is no guarantee that the customer would recycle the brace upon delivery and thus, this would have adverse environmental effects and violate Cisco’s commitment to sustainability. Increasing the sheet metal thickness is only marginally less sustainable.

Because the estimates for sustainability are rough, sustainability will be compared against each alternative solution in an analytical hierarchy process. Instead of classifying an external brace to add a certain amount of carbon to the environment each year, the sustainability of the brace can be rated to be significantly worse than an added bead to the product.

6.3.5 Solution Selection Method

Once each alternative was measured on each of its criteria (robustness, cost, implementation time, and sustainability), the qualities and alternatives were compared to each other in order to select alternative solutions and/or combinations of solutions for the final design of the short term and long term proposals.

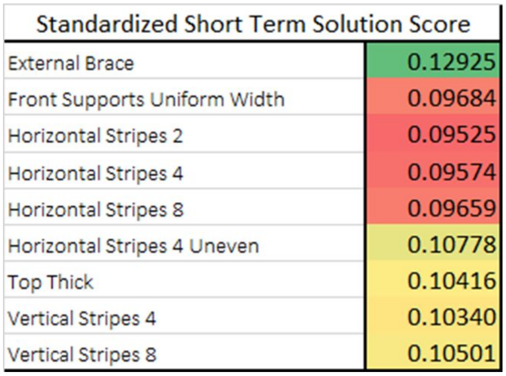
An analytical hierarchy process was used to compare the results and solution alternatives against one another. Two different AHP’s (Analytical Hierarchy Process) were used to evaluate the short term and long term solution.

Table 10 shows the standardized quality matrix and the averages of each quality row. Of the alternatives in this model, robustness had the heaviest weight followed by implementation time, cost, and lastly, sustainability.

	Short Term Qualities Standardized				
	Robustness	Cost	Sustainability	Implementation Time	Average
Robustness	0.687772926	0.690140845	0.409090909	0.746268657	0.633318
Cost	0.098253275	0.098591549	0.318181818	0.074626866	0.147413
Sustainability	0.076419214	0.014084507	0.045454545	0.029850746	0.041452
Implementation	0.137554585	0.197183099	0.227272727	0.149253731	0.177816

Table 10: Analytical Hierarchy Process Standardized Qualities for Short Term Solution

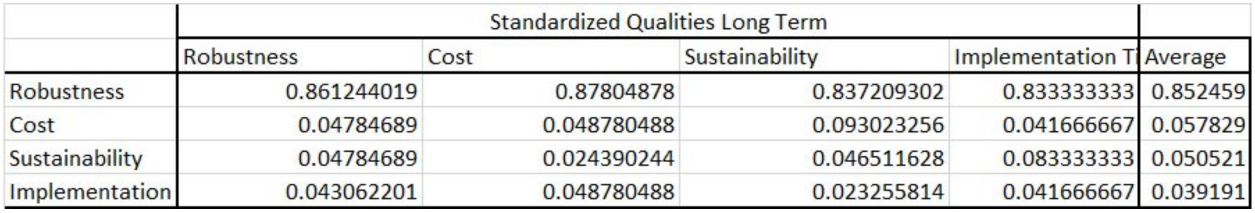
Table 11 shows the final standardized scores for the short term AHP for each design alternative. The external brace had the highest score of all of the alternatives, which is not surprising as it had the best robustness of any alternative and the fastest predicted implementation time, which were the two most important qualities in the evaluation.



Standardized Short Term Solution Score	
External Brace	0.12925
Front Supports Uniform Width	0.09684
Horizontal Stripes 2	0.09525
Horizontal Stripes 4	0.09574
Horizontal Stripes 8	0.09659
Horizontal Stripes 4 Uneven	0.10778
Top Thick	0.10416
Vertical Stripes 4	0.10340
Vertical Stripes 8	0.10501

Table 11: Standardized Scores of Short Term Solutions

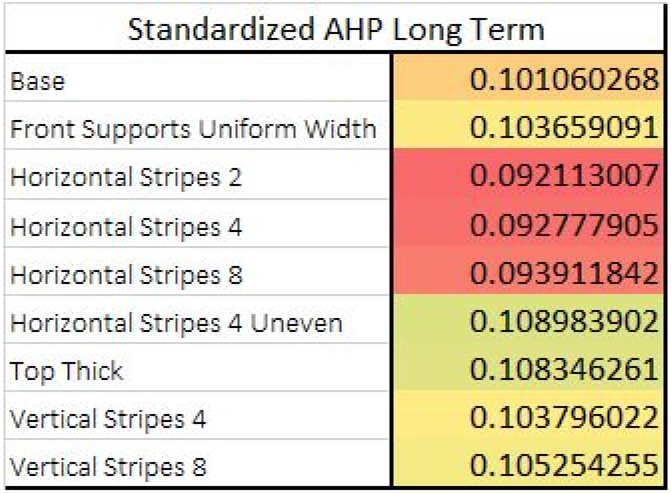
Table 12 shows the standardized quality matrix and the averages of each quality for the Long Term AHP. Robustness had the heaviest weight in this model followed by cost, sustainability, and implementation time.



	Standardized Qualities Long Term				
	Robustness	Cost	Sustainability	Implementation Ti	Average
Robustness	0.861244019	0.87804878	0.837209302	0.833333333	0.852459
Cost	0.04784689	0.048780488	0.093023256	0.041666667	0.057829
Sustainability	0.04784689	0.024390244	0.046511628	0.083333333	0.050521
Implementation	0.043062201	0.048780488	0.023255814	0.041666667	0.039191

Table 12: Standardized Qualities for Long Term AHP

Table 13 shows the final standardized scores for the long term AHP. Of the embossing methods, the uneven emboss was the best alternative. Even front supports and thicker top sheet metal also proved to be viable options at improving the robustness of the chassis. The external brace was not included in the study as it was not sustainable.



Standardized AHP Long Term	
Base	0.101060268
Front Supports Uniform Width	0.103659091
Horizontal Stripes 2	0.092113007
Horizontal Stripes 4	0.092777905
Horizontal Stripes 8	0.093911842
Horizontal Stripes 4 Uneven	0.108983902
Top Thick	0.108346261
Vertical Stripes 4	0.103796022
Vertical Stripes 8	0.105254255

Table 13: Standardized Alternative Values for Long Term AHP

Once the group of viable solutions was identified, several prototypes were modeled and tested in Solidworks; only the best one was chosen. While the AHP methodology is good at identifying the best of individual alternatives, it does little to predict the interactions between them.

6.4 Additional Experiments

An experiment was conducted to understand the effects the PCB had on the model. Because it is off set from the sheet metal and is rigidly constrained with connectors in the FEA model it adds a lot of stiffness to the bottom sheet metal. When the PCB was removed from the model it had it increased the stress in all areas on the top down drops and did not affect the model much (<1%) in the top up drops. This showed how the stiffness of the bottom sheet affected the model.

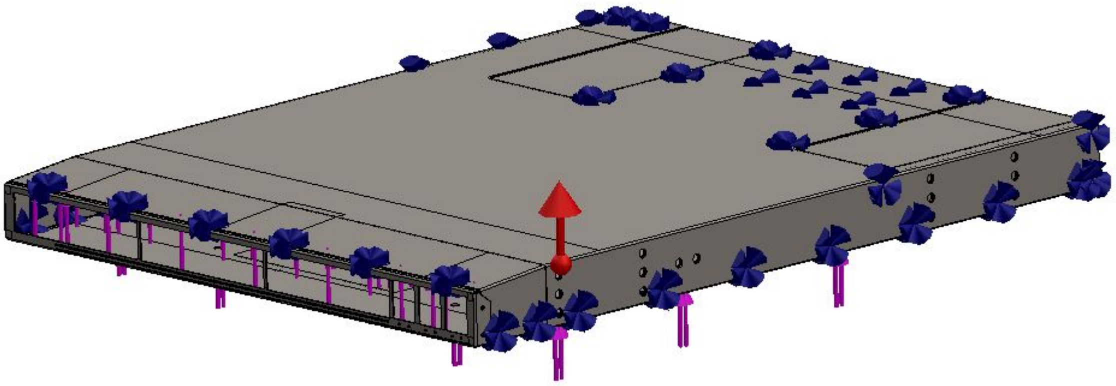


Figure 35: No PCB Fea Model

Another experiment that was conducted but did not factor into the final analysis was the effect that foam orientation has on the stresses in the model. Because the foam is what it is in contact with the chassis, it is the source where force is applied. The idea was that the chassis should be supported along the sides instead across the front and back of the chassis; the sides are much more robust than the front and back. The surface area in contact with the chassis was kept the same because in real life the surface area is critical to the amount of impact that the foam will absorb. To much foam would not allow the foam to condense and would transfer the load directly into the chassis and not enough would make it condense too much.

The foam orientation drastically reduced the stress across all the different sites. It reduced stress 50% on average on site 1 and 36% in sites 2-12. However, it should be noted that changing the orientation of the foam fundamentally changes the FEA model and thus these results can not be directly compared to the base model. For this reason it was excluded from the analysis.

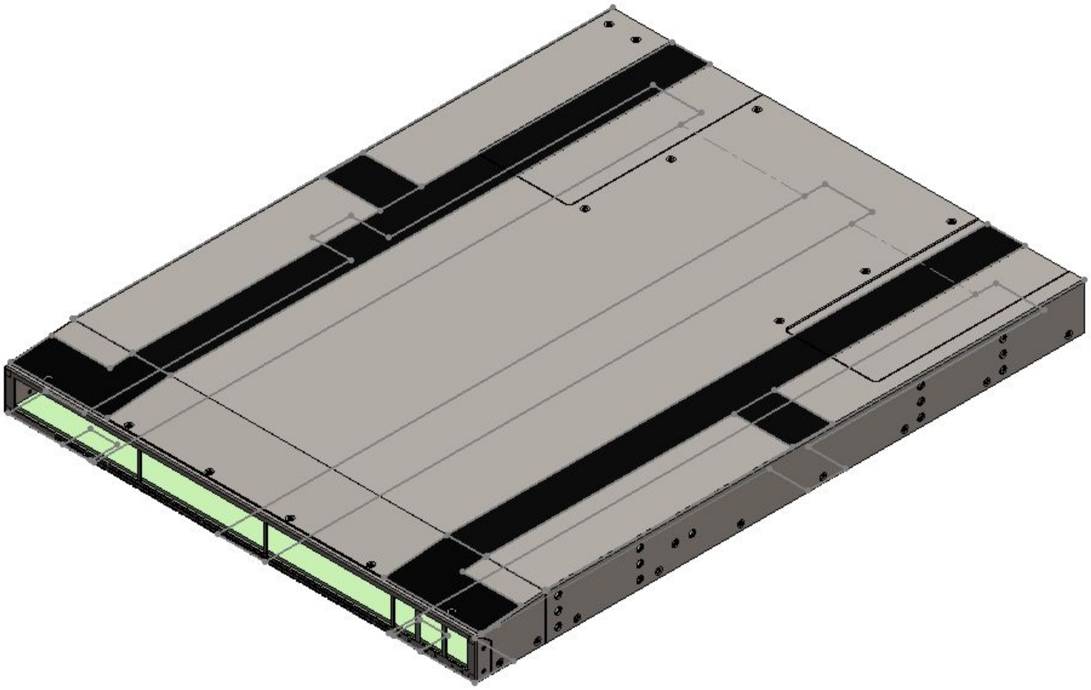


Figure 36: Foam Orientation FEA Model

Additional experiments that could be run to investigate the root cause of the bending damage seen in the Nexus 9200 top and bottom cover is thermal expansion analysis. This simulates how different material can contract or expand from assembly to the customer due to changing temperatures. What could potentially result from this is the contracting or expansion of the PCB or the bottom cover of the unit could lead to bending in the cover due to the screws fastened down. This idea was not tested by our team, as it did not directly relate to increasing the durability of the chassis, but was considered as an alternative experiment.

Basic procedure for this experiment would be to individually place the top cover, bottom cover and the PCB under a simulation where the temperature would be changed until a threshold value of deformation was observed in the part. This deformation would be measured at holes, the locations where the parts are screwed together. Upon finding this deformation the parts would be mated together, two at a time, and then ran under a simulation with the same change in temperature. Ideally, there would be a difference in the location of the holes for the screws for the two parts, resulting in a stress being pre loaded to a different location of the unit, specifically the cover. Deformation in these locations would be then measured and compared to the physical damage observed by customers.

7. Conclusions and Recommendations

There are two solutions. The first is a short term solution: a fix that can be applied as quickly as possible. The second is a long term solution: one that is more cost effective and sustainable, but would take longer to implement. Finally, future design recommendations were created based on the testing done in this project.

7.1 Short Term

7.1.1 Description of Solution

As previously mentioned, we wanted to design a short-term solution that could be implemented as quickly as possible upon noticing instances of unit damage. Criteria for this solution included something that would solve the issue but have a relatively fast, cheap, and painless implementation. This ruled out the possibility of adding any sort of embossing, which would require engineering change orders that take time to process. What we decided upon was selecting the external metal brace that would mount in holes along the sides of the chassis and sit upon the front cover. This external brace has three vertical beads in order to increase its stiffness and is made of 19 gauge AISI 1020 cold rolled steel. Based on our testing results, we saw that it adds enough rigidity for the chassis to sustain rough shipping environments and it should be noted that the chassis would be removed and discarded upon delivery. Figure 37 shows the 3D render of the chassis with the brace attached.

Figure 37: CAD Render of External Brace Attached to Chassis

7.1.2 Solution Justification

During testing, the external brace reduced loads in top-down on average 29% across sites 2-12 and 23% at site 1. It reduced the maximum stress by 29%. Amongst all of the alternatives, it was the most effective across all measures of robustness. Of all of the alternatives, it was estimated to be the fastest to implement. Figure 38 shows a comparison between an external brace and the base model across all sites on the top-down drop.

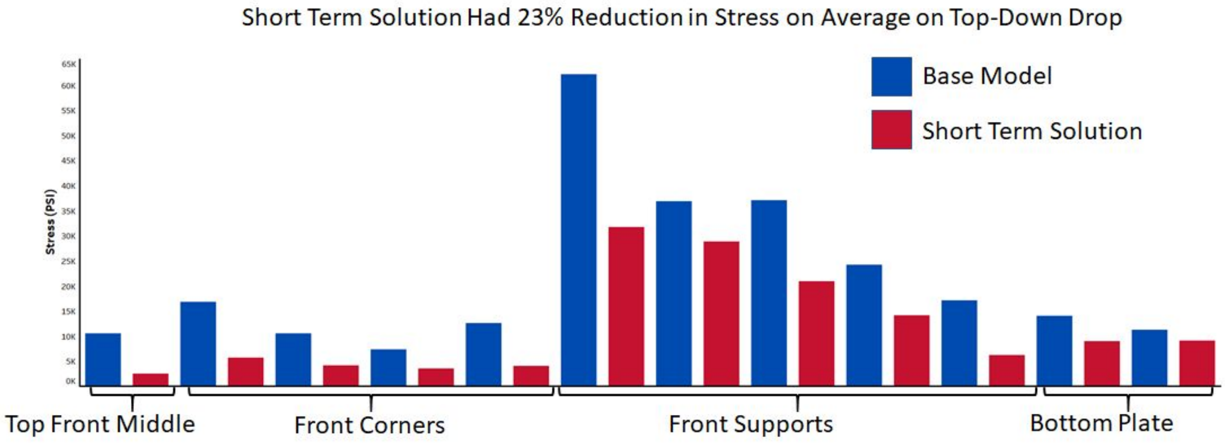


Figure 38: Top Down Drop Results: Base Model vs. Short Term Solution

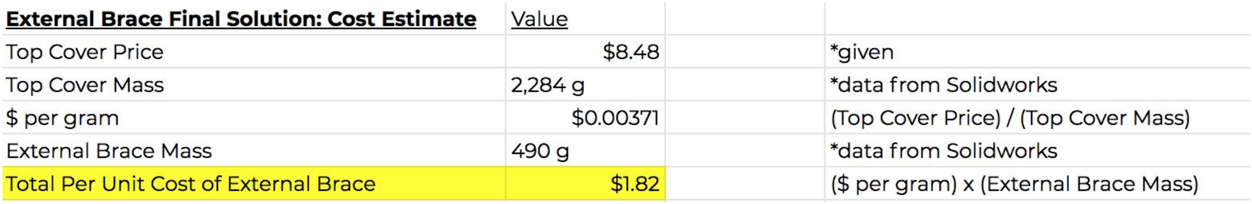
For the short term solution the external brace was explicitly chosen because of its effectiveness and most importantly its implementation time. The solution is relatively cheap at $1.82 per unit. Unfortunately, this design does not meet Cisco’s values for sustainably and thus should be used as a bandaid, temporary solution if necessary.

7.1.3 Implementation Plan

While new stamping dies would need to be designed and created to produce the brace, the brace's simple shape, simple design, moderate size, and lack of changes to the actual product would result in faster implementation times. We estimated the time to implement the brace to be around 2 months. The path to implementation starts with creating accurate prototypes and drop testing a chassis with a brace installed to ensure functionality of the current design. Then the stamping dies would be designed and machined, followed by testing the first few batches of braces and redesigning the brace and stamping dies if necessary. The Packaging Engineering team would need to conduct drop tests to ensure Cisco's design standards were met. Full production of the brace could begin shortly afterwards. This design would require changing the packaging foam to make room for the brace, these changes could be implemented in a number of weeks since the changes are minor.

7.1.4 Economics of Design

We estimate the total cost of implementing this solution to be $1.82 per unit. Using data that was provided to us - the weight and price of manufacturing the top cover of the current design - we calculated the per gram cost of sheet metal that Cisco pays. We then multiplied the per gram cost of sheet metal by the total weight of the brace to determine the total cost. Table 14 below provides a breakdown of the financial estimate.



External Brace Final Solution: Cost Estimate	Value		
Top Cover Price	$8.48		*given
Top Cover Mass	2,284 g		*data from Solidworks
$ per gram	$0.00371		(Top Cover Price) / (Top Cover Mass)
External Brace Mass	490 g		*data from Solidworks
Total Per Unit Cost of External Brace	$1.82		($ per gram) x (External Brace Mass)

Table 14: Breakdown of External Brace Cost Estimate

Considering the fact that the current state of the issue adds approximately $8.30 per unit, it is financially sensible to invest this $1.82 per unit in order to prevent any damages from occurring at all.

7.1.5 Conformance to Standards

Our FEA simulation was modeling an ASTM D5276 simulated drop test. It was key that our simulation conformed to a standard drop test as best possible within FEA. Since drop testing is down when a product is in a package, to simulate a drop during shipping, we needed to include the packaging in our simulation. We did so by placing fixtures on the top and bottom covers in the location of the current Nexus 9200 packaging. These fixtures were set with an elastic modulus of 10.66 lb/in^2 to mimic the cushion the foam provides.

Sample Size of 3	Taking multiple samples was not possible in FEA since results will be the same
Drop height of 36 inches	30 G's of force was our estimated force applied to the package from a drop height of 36 inches. The Cisco package testing reports were performed from a drop height of 36 inches for top and bottom drops. 30 G's was roughly half of the measured results for these tests.
Drop Orientations	Defined to be on the top and bottom of the package. Standard drop testing is performed on all sides, corners and edges.
Testing Temperature	To be performed at standard room temperature. This was not a possible defining parameter in SolidWorks.

Table 15: Basic Parameters of ASTM D5276 Standard

7.2 Long Term Solution

The long term solution addresses any redesigns of the chassis that can take place after the issue has been temporarily resolved. Overall, it is cheaper, more effective, and more sustainable than the short term solution. However, it takes considerably longer to implement due to product redesigns that require extensive reigneering, testing, and manufacturing changes.

7.2.1 Description of Solution

As a long term fix, we wanted an effective design modification that would not only prevent the issue from occurring but also be sustainable and relatively cheap to implement. The solution design that we propose involves three parts that each contribute to making the chassis more durable. First, we want to add two 1.5 inches long emboss stripes along the top sheet metal cover. After experimenting and testing over twenty different emboss configurations, the two uneven stripes gave us the best and most feasible results. The stripes would start along the front right of the chassis and serve two purposes: to distribute stress away from the crucial front supports as well as add overall sturdiness to the cover. Furthermore, we want to increase the sheet metal thickness of the top cover to 19 gauge AISI 1020 cold rolled steel. Increasing the sheet metal thickness makes the cover less susceptible to bending and 19 gauge was chosen because it does not add overly excessive height to the 1RU, allowing it to retain its ability to slide in to server racks. Our last design modification is to evenly space out the front supports of the chassis. Given that the front supports see the highest amounts of stress in the current design, spacing the front supports allows stress to be distributed amongst them. Images of the final design can be found below.

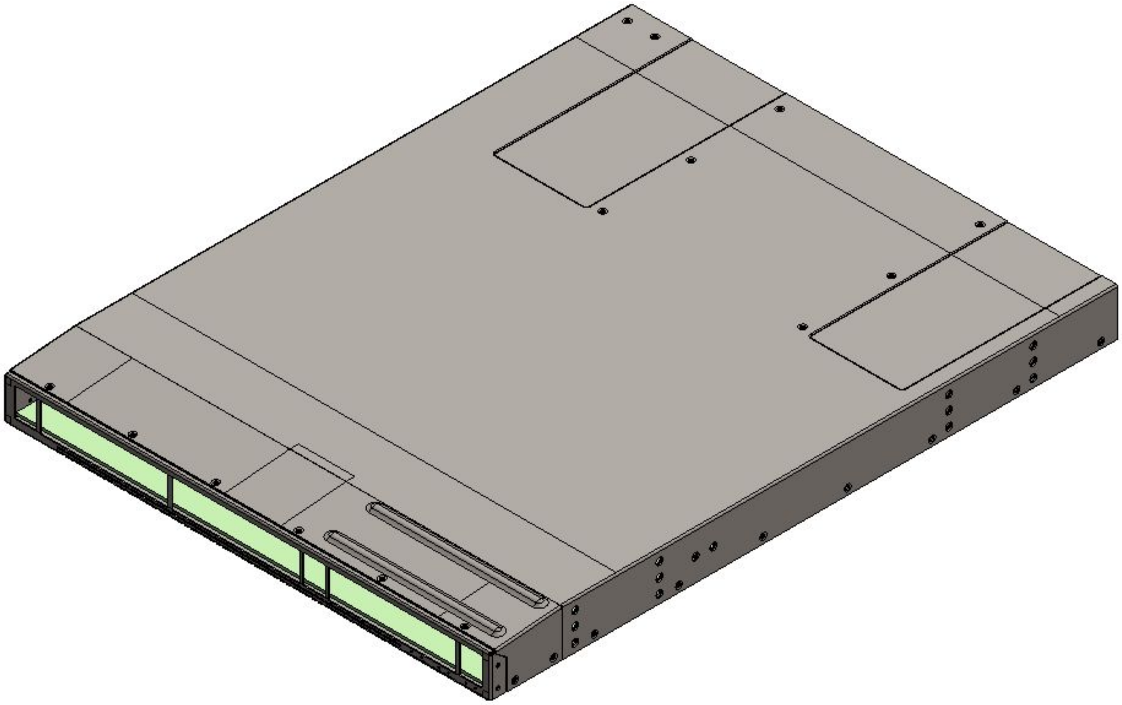


Figure 39: CAD Render of Long Term Solution Design

7.2.2 Solution Justification

During testing, the external brace reduced loads in top-down on average 63 % across sites 2-12 and 65% at site 1. It reduced the maximum stress by 61%. Figure 40 shows a comparison between the longterm design in blue and the base model in grey across all sites on the top-down drop.

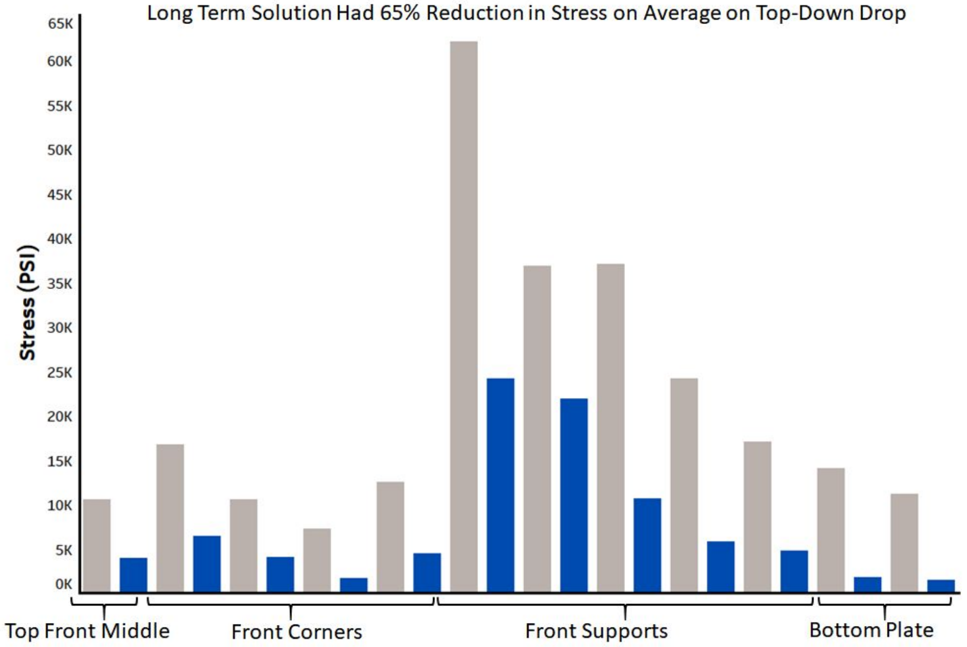


Figure 40: Top Down Drop Results: Base Model vs. Long Term Solution

For the long term solution the external brace was explicitly chosen because of its effectiveness and most importantly its implementation time. The solution is relatively cheap at $1.32 per unit.

7.2.3 Implementation Plan

Changing the thickness of the sheet metal significantly increases the implementation time of this plan as it requires redesigning the product to ensure all design criteria are still met as well as redesigning and machining the new stamping dies. The size and complexity of the part and die exponentially increase the time required, we estimated 4 months to fully implement. The path to implementation starts with creating accurate prototypes and drop testing a chassis with a redesigned top cover installed to ensure functionality of the current design. Then the stamping dies would be designed and machined, followed by testing the first few batches of tops and redesigning the top and accompanying stamping dies if necessary. The Packaging Engineering team would need to conduct drop tests to ensure Cisco's design standards were met. Full production of the new top could begin shortly afterwards. These changes would not require any modifications of the current packaging design, speeding up implementation.

7.2.4 Economics of Design

We estimate the total cost of implementing this solution to be $1.32 per unit. Using data that was provided to us - particularly the weight and price of manufacturing the top cover of the current design - we calculated the per gram cost of sheet metal that Cisco pays. We then multiplied the per gram cost of sheet metal by the total weight of the thicker top cover to determine the total estimate. As previously mentioned in Section 6.3.2, the costs of adding beading and rearranging the front supports were estimated as $0 per unit. Table 16 below provides a breakdown of the financial estimate for this solution.

Thick Sheet Metal: Cost Estimate	Value		
Top Cover Price	[redacted]		*given
Top Cover Mass	[redacted]		*data from Solidworks
$ per gram	[redacted]		(Top Cover Price) / (Top Cover Mass)
Top Cover Mass	[redacted]		*data from Solidworks
Top Cover Cost: Thicker Sheet Metal	[redacted]		($ per gram) x (Top Cover Mass)
True Cost of Changing Thickness	$1.32		Thick Top Cost - Current Top Cost

Table 16: Breakdown of Thicker Sheet Metal Cost Estimate

7.3 Recommendations

Our data shows that across all the alternatives the most critical regions on the chassis are the front supports. This being said, design choices should revolve around reducing the stress on the parts of the chassis. Increasing the rigidity of the top and bottom, braces, and distributing the supports are all effective ways of doing this.

8 Future Directions

This section discusses project next steps as well as improvements that could have been made to increase the accuracy of the results.

8.1 Implementation of Results

The next step towards implementation would be to create prototypes of the brace and the redesigned top, drop them according to Cisco's testing standards, and analyze the results. Upon passing, new stamping dies would need to be designed, machined, and tested. The first few batches of the new dies would need to be carefully scrutinized for fit and errors. Once the dies are operational, they would swap places with the current dies and go into production. This process would be much slower for the redesigned top as it would require time consuming ECO's (Engineering Change Orders) and the dies would take a lot longer to design and make. Our comprehensive analysis and testing makes us confident that the bending issue will not return if our solutions are implemented.

As Cisco has already eliminated the problem as stated earlier, implementing either of our solutions would result in unjustified costs that bring little to no benefit. If the bending problem was to start happening again, our solutions are worth considering as they address the problem at hand as well as speed, cost, and sustainability.

8.2 Improvements to Project

There are several improvements that could be made to this project to enhance the accuracy of the results and reveal innovation solutions. They are mainly through FEA.

The accuracy of the FEA model could be greatly improved. Firstly, a different meshing method could have been used to more accurately represent real life .Unfortunately Solidworks is very limiting on the meshing it will allow the user to do. Generally speaking, thin sheet metal parts should be modeled as shells.

As mentioned before the foam is not modeled well in this simulation. Running a nonlinear dynamic study of the model would provide better and more accurate results.

Finally, a full factorial testing setup could have been used to study the interactions between design alternatives.

9. Project Analysis

This section depicts the specific plan and vital steps taken to ensure our project conclusions complete to Cisco's specifications and delivered on time.

9.1 Project Management Analysis

Steps taken to keep our project on track and have our final report delivered on time were setting due dates for our team. Examples of this would be defining individual tasks for group members by April 14th and vet our solution by May 10th.

Break up of individual assignments was done as so. Cole, was responsible for model validation using Roark's Formulas for Stress and Strain, Colin designed and manufactured the external brace which was used a short term solution. He also led the team in modeling of our solution designs. Ethan led the design and simulation of our FEA model in SolidWorks so that all constraints were met. Rahul gathered cost information from Cisco and provided the economic analysis on the benefits or detriments of each proposed solution.

9.2 Client Communication Analysis

The quality of our communications with Cisco have increased greatly as the project progressed, as we were better equipped to handle the next set of goals. The vast majority of communication was through video conferencing, allowing us to have face-to-face discussions with the Test & Quality, Packaging, and Engineering teams. After each meeting, a summary was sent out providing a record of the discussions, decisions made, and guidance given. We referenced these summaries many times, helping us avoid mistakes in our assumptions when designing new solutions. Table 17 lists our meetings with Test & Quality, Packing, and Engineering. While email was used extensively in this project, we felt that the record of our meetings paints a better picture of our interactions with the client.

9.3 Teamwork Analysis

Teamwork analysis will be measured by individuals commitment to group meetings and involvement in group discussion.

First, group meetings were held weekly in class and before and after client meetings. Group members were present for nearly every meeting with our team and with our sponsor. If there was an absence it was always communicated with the team prior to the event they would miss. When an individual of the group felt it was necessary for the team to meet, a team meeting would be called. When this occurred we continuously saw commitment from every team member to show up and contribute to that days specific goal, unless a valid excuse was given.

Second, meetings were being conducted it was expected that all members listen and contribute to completing the daily task. Some days that task would be consulting on fixture design with Dr. Peter Schuster, other days it would be designing new models within SolidWorks. Regardless or what was asked of the team all members worked in the best way they could to meet that daily goal.

10 Bibliography

The cited sources below are cumulative of all the sources used within the entire report.

10.1 Sources

Protective Packaging for Distribution: Design and Development, by Daniel Goodwin and Dennis Young, DEStech, 2011.

Lee, Henry. "Introduction to Finite Element Analysis & Tools." Cisco. 2019.

Roark, Raymond J., et al. Roark's Formulas for Stress and Strain. Seventh ed., McGraw-Hill, 2002.

Cast Study: XXXXXXXX 1RU. Cisco, 2019, *Cast Study: XXXXXXXXX 1RU*.

Earl, James, and Mike Keys. *Package Test Report*. Cisco, 2017, *Package Test Report*.

REFERENCES (or BIBLIOGRAPHY)………………………………………….…..…..
APPENDICES (if appropriate)

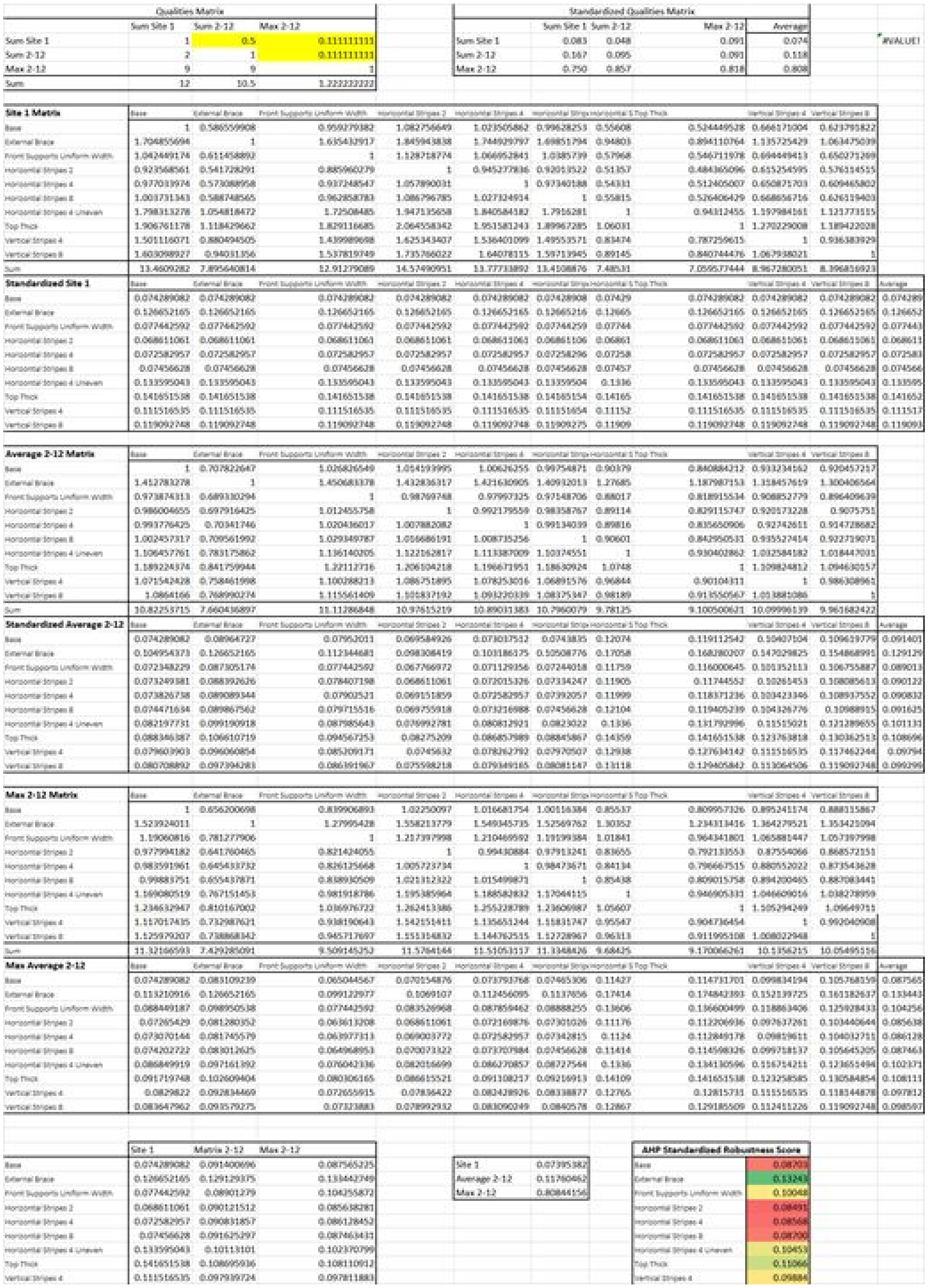
1. Documented communication with the client organization, including at least 2-3 visits. A table of dates, types, and highlights of each communication (from emails, calls, and visits)

Date	Type	Topic	Teams	Attendance
1/31/19	Meeting	First Project Meeting, Understanding Problem	T&Q	Christine Nolan-Brady
2/14/19	Meeting	Understanding Problem	T&Q	Christine Nolan-Brady
2/19/19	Meeting	Questions with Packaging	T&Q, Packaging	Christine Nolan-Brady, Mark Doutt, James Earle
3/5/19	Meeting	Weekly Meeting	T&Q	Christine Nolan-Brady
3/6/19	Meeting	Questions with Product Engineering	T&Q, Engineering	Christine Nolan-Brady, Harvey Yang
3/13/19	Meeting	Weekly Meeting	T&Q	Christine Nolan-Brady
3/18/19	Meeting	Project Update	T&Q	Christine Nolan-Brady
5/7/19	Meeting	Review FEA model with Cisco	T&Q, Engineering	Christine Nolan-Brady, Henry Lee

5/14/19	Meeting	Review updated FEA model	T&Q, Engineering	Christine Nolan-Brady, Henry Lee
5/17/19	Meeting	Final project presentation to Test & Quality, Packaging, Product Engineering	T&Q, Packaging, Engineering	Christine Nolan-Brady, Rob Grimes, Greg Twiss, Bob Loose, Mark Doutt, Robert Lee, Harvey Yang, James Earle

Table 17: Meetings

2. AHP Robustness



Qualities Matrix	Sum Site 1	Sum 2-12	Max 2-12
Sum Site 1	1	0.5	0.111111111
Sum 2-12	2	1	0.111111111
Max 2-12	9	9	1
Sum	12	10.5	1.222222222

Standardized Qualities Matrix	Sum Site 1	Sum 2-12	Max 2-12	Average
Sum Site 1	0.083	0.048	0.091	0.074
Sum 2-12	0.167	0.095	0.091	0.118
Max 2-12	0.750	0.857	0.818	0.808

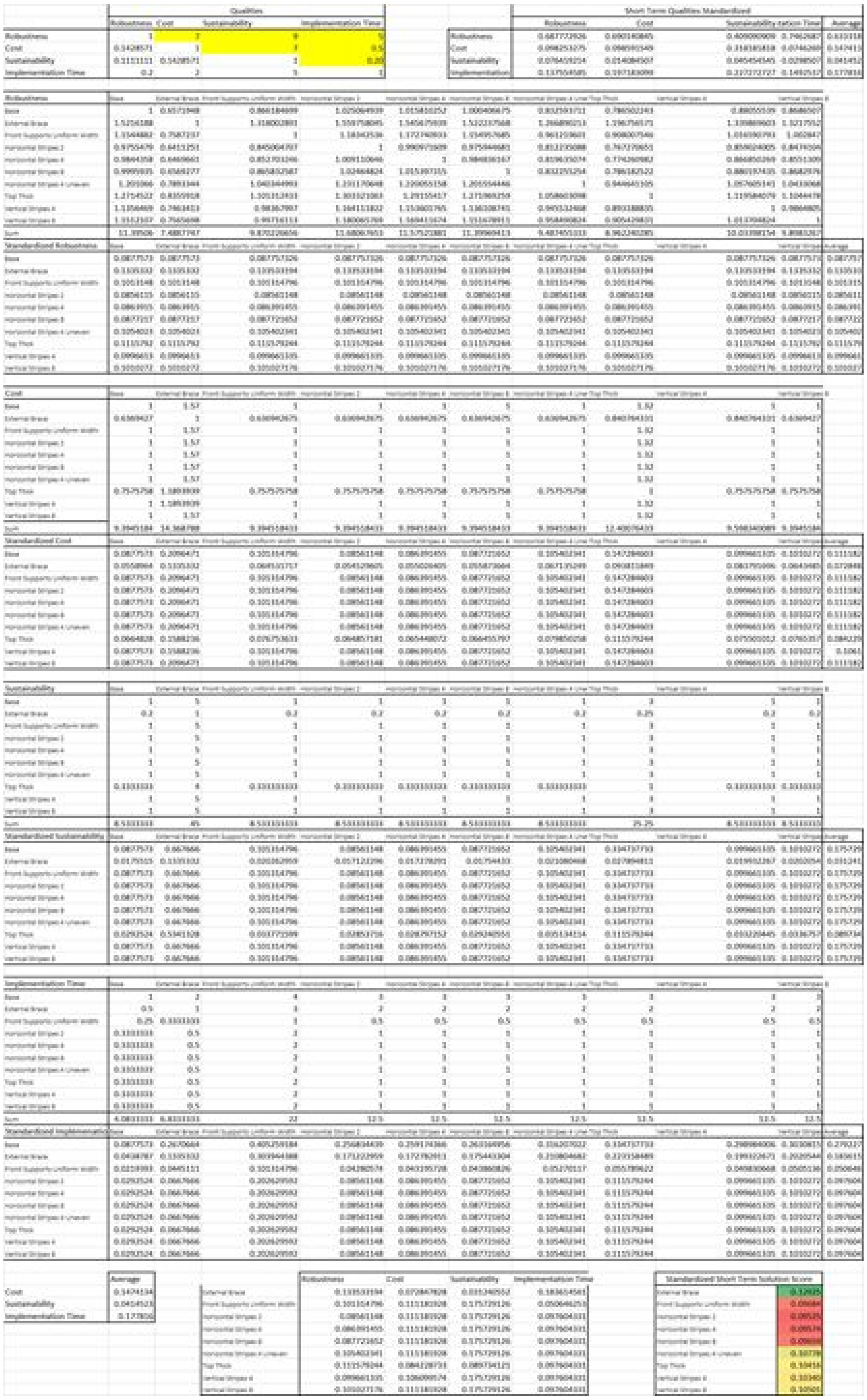
#VALUE!

	Site 1	Matrix 2-12	Max 2-12
Base	0.074289082	0.091400696	0.087565225
External Brace	0.126652165	0.129129375	0.133442749
Front Supports Uniform Width	0.077442592	0.08901279	0.104255872
Horizontal Stripes 2	0.068611061	0.090121512	0.085638281
Horizontal Stripes 4	0.072582957	0.090831857	0.086128452
Horizontal Stripes 8	0.07456628	0.091625297	0.087463431
Horizontal Stripes 4 Uneven	0.133595043	0.10113101	0.102370799
Top Thick	0.141651538	0.108695936	0.108110912
Vertical Stripes 4	0.111516535	0.097939724	0.097811883

Site 1	0.07395382
Average 2-12	0.11760462
Max 2-12	0.80844156

AHP Standardized Robustness Score	
Base	0.08703
External Brace	0.13243
Front Supports Uniform Width	0.10048
Horizontal Stripes 2	0.08491
Horizontal Stripes 4	0.08568
Horizontal Stripes 8	0.08700
Horizontal Stripes 4 Uneven	0.10453
Top Thick	0.11066
Vertical Stripes 4	0.09884

3. AHP Short Term



4. AHP Long Term