CNC Milling Bracket Design Report

June 4th, 2023

Valeria Castro M, Tadeo Carrillo R, Gabriela Oyarzún B.

Version 1 Overview:

During the first lab part of the CNC Bracket Design Project, we generated an initial model that can be seen below:

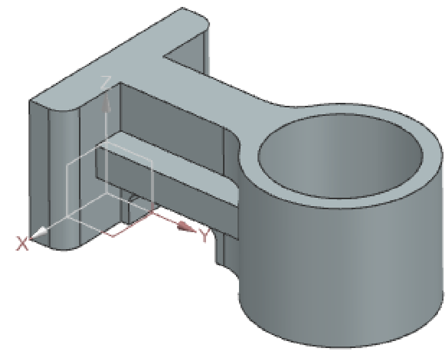


Figure 1: Initial Bracket Design V1

While working on this first design, we wanted to know where we had the highest stress concentrations and where our part would deflect most. To do this, we used two methods: hand calculations and Finite Element Analysis (FEA).

To do hand calculations, we first solved for the reaction forces that would be present in our bracket assembly with the spindle to get a grasp of the loads it was going to be subjected to:

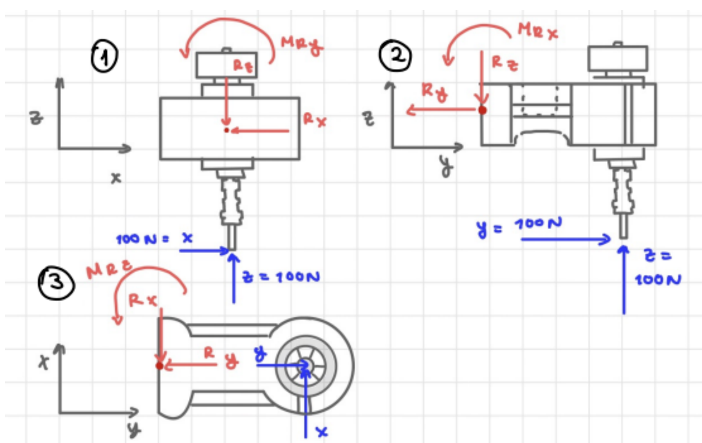
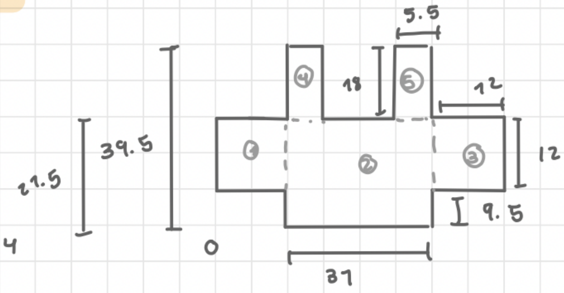


Figure 2: Reaction Forces FBD.

We used these forces as a reference for our initial hand calculations on stress and deflection for the initial design of the bracket. We started by taking a cross-section of our design and using it to calculate these values:

ASSIGNMENT 2



Step 1

① $I_{x_1} = \frac{1}{12}(12)(12)^3$

$= 1728 \text{ mm}^4$

② $I_{x_2} = \frac{1}{12}(31)(21.5)^3$

$= 25674.14 \text{ mm}^4$

③ $I_{x_3} = 1728 \text{ mm}^4$

④ $I_{x_4} = \frac{1}{12}(5.5)(18)^3$

$= 2673 \text{ mm}^4$

⑤ $I_{x_5} = 2673 \text{ mm}^4$

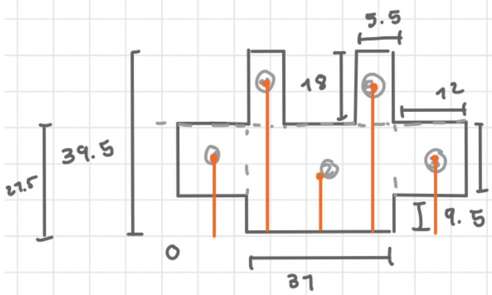
Step 2

$$z_c = \frac{A_1 z_1 + A_2 z_2 + A_2 z_3}{A_1 + A_2 + A_3}$$

$A_1 = 144 \text{ mm}^2$ $A_4 = 99 \text{ mm}^2$

$A_2 = 666.5 \text{ mm}^2$ $A_5 = 99 \text{ mm}^2$

$A_3 = 144 \text{ mm}^2$



$z_1 = 9.5 + 12/2 = 15.5 \text{ mm}$

$z_2 = 21.5/2 = 10.75 \text{ mm}$

$z_3 = 15.5 \text{ mm}$

$z_4 = 30.5 \text{ mm}$

$z_5 = 30.5 \text{ mm}$

$$z_c = \frac{17667.88}{1152.5} = 15.33 \text{ mm}$$

Step 3

$z_1 = z_c - z_1 = 0.17 \text{ mm}$

$z_2 = 4.58 \text{ mm}$

$z_3 = 0.17 \text{ mm}$

$z_4 = z_4 - z_c = 30.5 - 15.33 = 15.17 \text{ mm}$

$z_5 = 15.17 \text{ mm}$

Step 4

$I_{x1c} = I_{x1} + A_1 z_1^2$

$= 1728 + (144)(0.17)^2$

$= 1732.16$

$I_{x2c} = I_{x2} + A_2 z_2^2$

$= 25674.14 + (666.5)(4.58)^2$

$= 39654.97$

$I_{x3c} = 1732.16$

$I_{x4c} = 2673 + (99)(15.17)^2$

$= 25455.76$

$I_{x5c} = 25455.76$

Step 5

$$I_{total} = 94030.75 \text{ mm}^4 \qquad z_c = 15.33 \text{ mm}$$

$$z = 39.55 - 15.33 = 24.22 \text{ mm}$$

F(mm)

$$\sigma = \frac{Mc}{I} = \frac{10 \text{ N}\cdot\text{m}\,(24.22 \cdot 10^{-3})}{9.403 \cdot 10^{-8}} = 3.1075 \cdot 10^{6} \text{ Pa}$$

$$= 3.1075 \text{ MPa}$$

$$\frac{FL^3}{EI} + \frac{FL}{EA} = 5.74856 \cdot 10^{-4} \text{ m} + 4.6402$$

$$= 0.57486 \text{ mm}$$

Figure 3 and 4: Stress and Deflection Calculations

As shown, our calculated stress was about 3.1075 MPa and our calculated deflection was of about 0.575 mm. Then we proceeded to run FEA on our part gathering the following results:

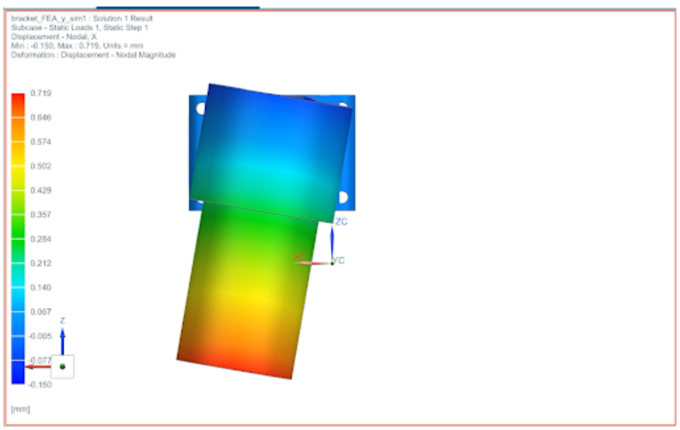


Figure 5: Displacement in the X direction

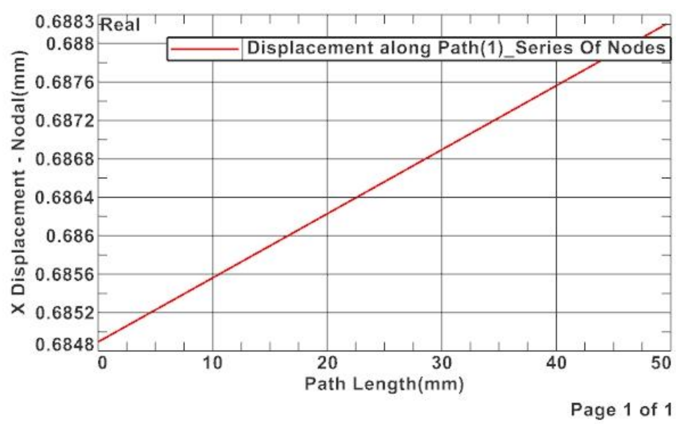


Figure 5: Displacement Line Plot in the X

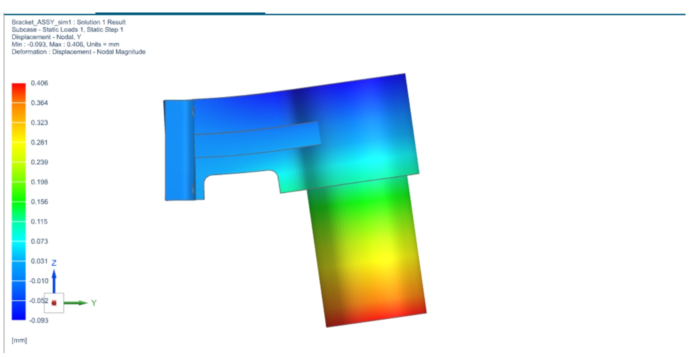


Figure 7: Displacement in the Y direction

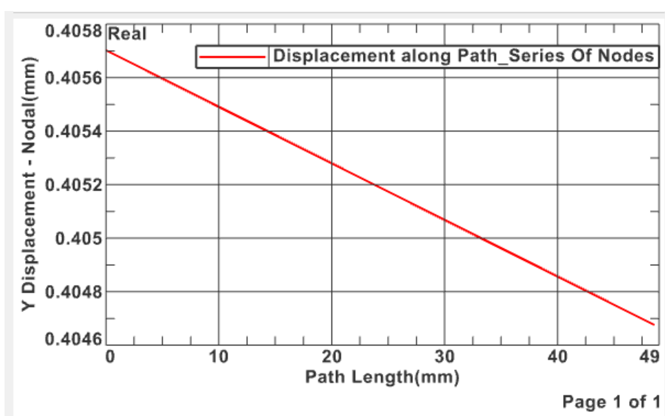


Figure 8: Displacement Line Plot in the Y

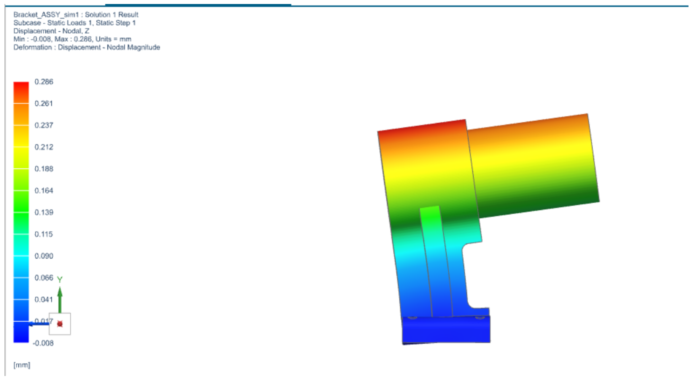


Figure 9: Displacement in the Z direction

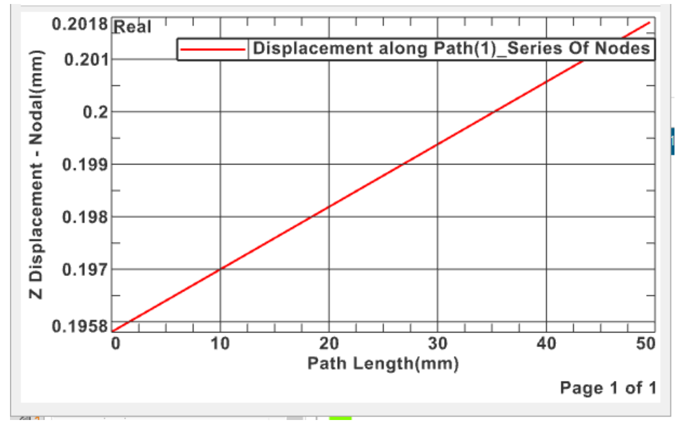


Figure 10: Displacement Line Plot in the Z

Based on our FEA, we made changes to our design. We took into account that there was stress concentration at the intersection of the base and the neck of the bracket. This value indicated instability for which we decided to incorporate fillets. Additionally, the most deflection occurs at the cylindrical spindle hole which indicated a critical bending point for which we wanted to add extra material for support. We also thought about adding small extrusions on the spindle hole to be able to use screws and nuts to fasten and secure the spindle.

Taking all of this into consideration, we decided to add to move forward with the following final design:

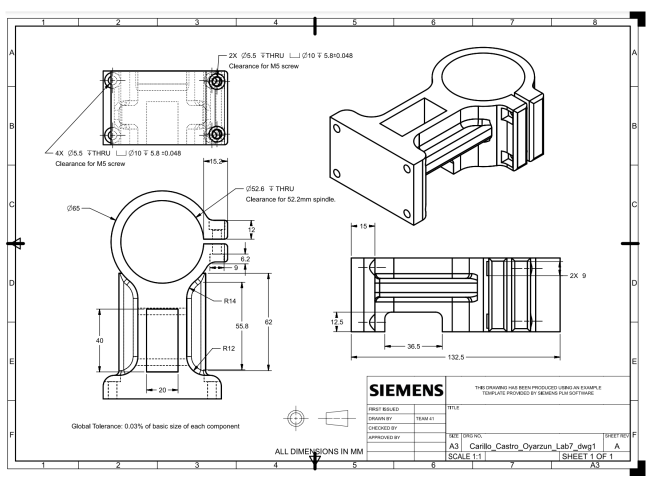


Figure 10: Final Design V1, engineering drawing.

Then, in our next lab, we tested our bracket, and we got the following results:

	Theoretical Applied Force (N)	Predicted Displacement from FEA (mm)	True Applied Force (N)	Actual Displacement (mm)	Difference in Predicted v Actual Displacement (mm)
X Direction	100	Min= 0.150 Max= 0.719	103 N	1.506 - 0.9 (as stated in the announcement) = 0.606	0.181
Y Direction	100	Min= 0.093 Max= 0.406	*not tested	*not tested	*not tested
Z Direction	100	Min= 0.008 Max= 0.286	105 N	0.993	-0.707

Table 1: Theoretical and Tested Bracket Data

On the other hand, our bracket weighed 204.2 g.

<u>Version 2 Overview:</u>

After testing our initial bracket design, we realized that putting support in the middle of our bracket wouldn't do much because that's where the neutral axis of our part is located. Instead, we decided to put a support on the top to reinforce our design and have better numbers on the deflection calculations. We also decided that we wanted it to be lighter, so we decided to make the subtraction in the middle of our bracket deeper. To lose even more material, we also decided to make the bottom curved and filleted, as shown in the figure below.

Here, we see in our FEA analysis that our predictions were wrong and our displacement calculations got larger:

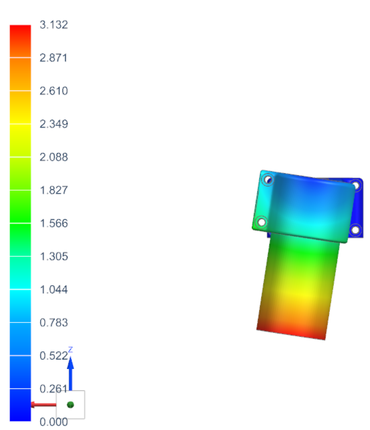


Figure 11: Displacement Heatmap in the X

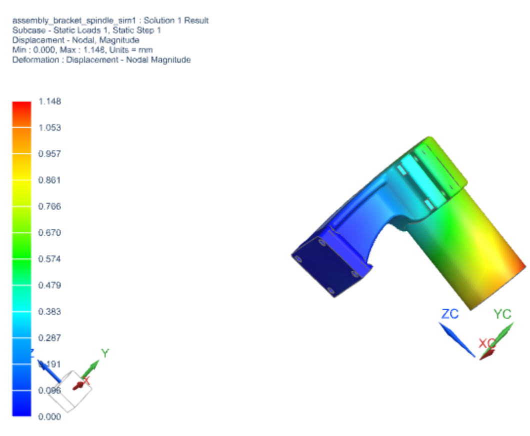


Figure 12: Displacement Heatmap in the Y

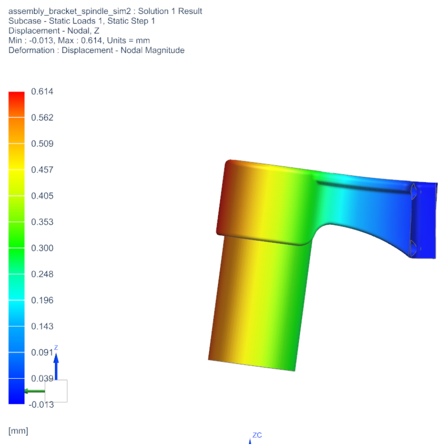


Figure 13: Displacement Heatmap in the Z

Because of this, we decided to redesign the bracket:

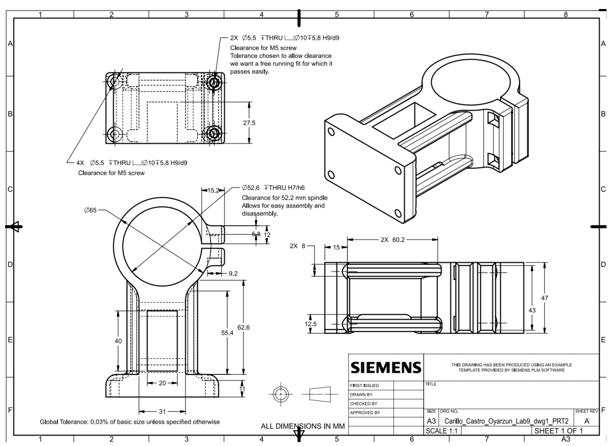


Figure 14: Drawing of Design V2

We decided to stick with our original design V1, which already had a good FEA, and added the support bars at the top and the bottom instead of the middle. Our testing results were the following:

Weight: 193 g.

Table 2: X-direction deflection:

	X_1 REAR	X_2 FRONT
0N	A_1 = 5.81	B_1 = 1.25
~100 N 105 (record actual force)	A_2 = 6.95	B_2 = 4.16
Actual End Displacement: $Displacement = (B_2 - B_1) - (A_2 - A_1)$		2.91−1.14 = 1.77

Table 3: Z-direction deflection:

	Z_1	Z_2
0N	A_1 = 5.4 mm	B_1 = 3.34 mm
~100 N 102N (record actual force)	A_2 = 6.10 mm	B_2 = 4.68 mm
Actual End Displacement: $Displacement = (B_2 - B_1) - (A_2 - A_1)$		1.34−0.7 = 0.64mm

Here we can see that our actual displacements are on spec (2mm>). And that the latter design is lighter than the original by about 10 gr. Making this a suitable design for the requirements stated at the start of the lab, even though we had to compromise on the x-deflection.

Looking back at our design process, we made our decisions based on the FEAs and what we learned in class. To improve our design further, we believe that we would have had to make more mindful decisions based on more careful observations on where we had to put more material for support (e.g. use our shear and normal stress distribution analysis) and where we could have taken material out to optimize weight. We also could have used a lower factor of safety to make our math align while not increasing the weight.

Future Bracket Version:

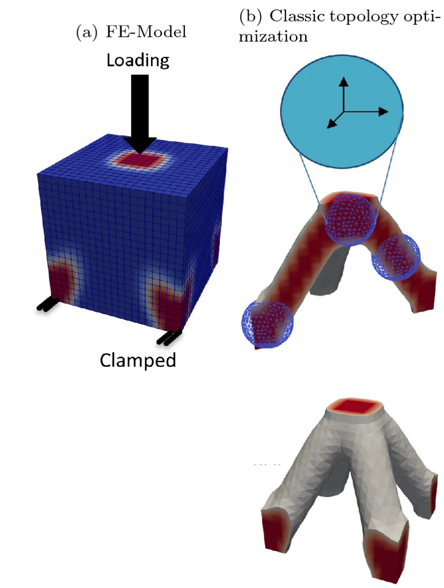


Figure 15: Visual model of classical topology optimization, based on a point-load. Adapted from source [1].

1. Propose potential improvements or modifications to the geometry of your part that could further enhance the bracket design, while still meeting the prescribed design specifications. Would you consider undoing any changes that you made to your V2 bracket?

Applying a topology-oriented approach to optimizing material and using iterative FEA seems like the simplest way to effectively optimize the weight and geometry of the part.

Topology is the study of solid surfaces and their continuity (see Figure 15 left). A load is applied at the top of the cubic structure and is dissipated among the 4 supports at the base of the cube as a continuous body[1]. With a specified tolerance, material that does not contribute to the transmission of force within that specified tolerance is eradicated.

Our part could be optimized with the specified techniques of topology and FEA in various ways. An example of where we could have used topology is the bottom cross-section of the bracket, which could be made in an optimal shape instead of a rectangle cutout. Furthermore, we could have seen if there where any areas where we could use a truss system to only target critical points of stress. Additionally, we could have used iterative FEA to control the area of most deflection (the intersection of the neck and the spindle mount) and add more material in the areas necessary.

2. Please explain how the loading conditions on this bracket differ during normal operation of the milling machine, from the static loading test that we conducted. What other loading conditions would a milling tool see during use?

The loading conditions on ths bracket differ during normal operation because the spindle will be subjected to multiple loads in different directions at once instead of one load and one direction at one time (x, y, or z). Additionally, instead of a constant load, the mill would be subjected to an impact load, which is the force provided by a sudden and quick load on a structural part. Aside from the static loading that we conducted, there are other forces acting on the spindle that we could have accounted for like vibrations or torques.

---

[1] This could be considered the visualization of internal stress, as it is—in a way—the transmission and dissipation of force from discrete body to discrete body based on surface to surface contact.

3. In considering these loading types, what are other changes to the design that might support this function for the lifetime of the part? You may consider altering some of the constraints that were given to you (e.g. materials selection).

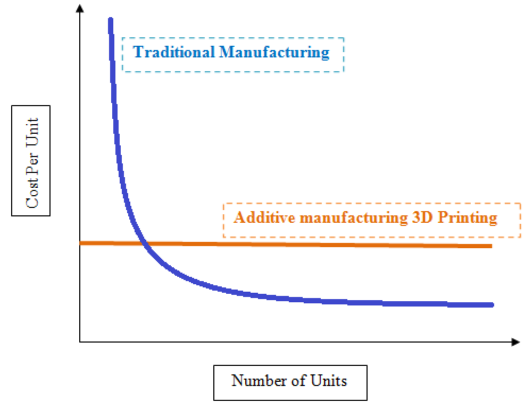


Figure 16: Graph showing the difference in cost trends when comparing tradition manufacturing and 3D Printing. Adapted from Source [2].

Another aspect of designing a future bracket should be material and manufacturing considerations. Nylon-12 in 3D printing has a tensile strength of 50MPa and a Young's Modulus of 1850 MPa. When compared to common manufacturing metals, these numbers are minuscule. Take Aluminum 6061 as an example; it has tensile strength of 276 MPa and a Young's modulus of 69 GPa. Both of these numbers make Nylon-12 appear incredibly weak, however, there is the great advantage of manufacturability. Making topologically sound Aluminum brackets has high CNC costs at a small quantity, whereas 3D printing Nylon-12 brackets has a consistently low cost.

4. How might incorporating these changes result in further changes to the geometry of the part?

Incorporating the changes in manufacturing and materials might allow for different geometry. While with the 3D printing we have the nozzle diameter as the smallest positive feature that can be made, the diameter on the CNC mill indicates the smallest negative feature possible [3]. Additionally, the CNC generally has a smoother surface finish than the 3d printing and also has tighter tolerances. In regards to how the material affects the weight of our part, we would need to design a bracket with less material and less volume to guarantee a light part. Steel could offer less deflection but the design process must prioritize weight optimization to able to provide a more effective part.

Sources

[1] N. Gerzen, T. Mertins, and C. B. W. Pedersen, “Geometric dimensionality control of structural components in topology optimization,” Struct Multidisc Optim, vol. 65, no. 5, p. 160, May 2022, doi: 10.1007/s00158-022-03252-7.

[2] R. Handal, “Additive Manufacturing As A Manufacturing Method,” JOSCM, vol. 10, no. 2, pp. 18–31, 2017, doi: 10.13140/RG.2.2.35883.90406.

[3] Markforged, "CNC vs 3D Printing: Understanding the Differences," [Online]. Available: https://markforged.com/resources/cnc-vs-3d-printing#:~:text=Geometry,feature%20that%20can%20be%20produced. [Accessed: Month Day, Year].