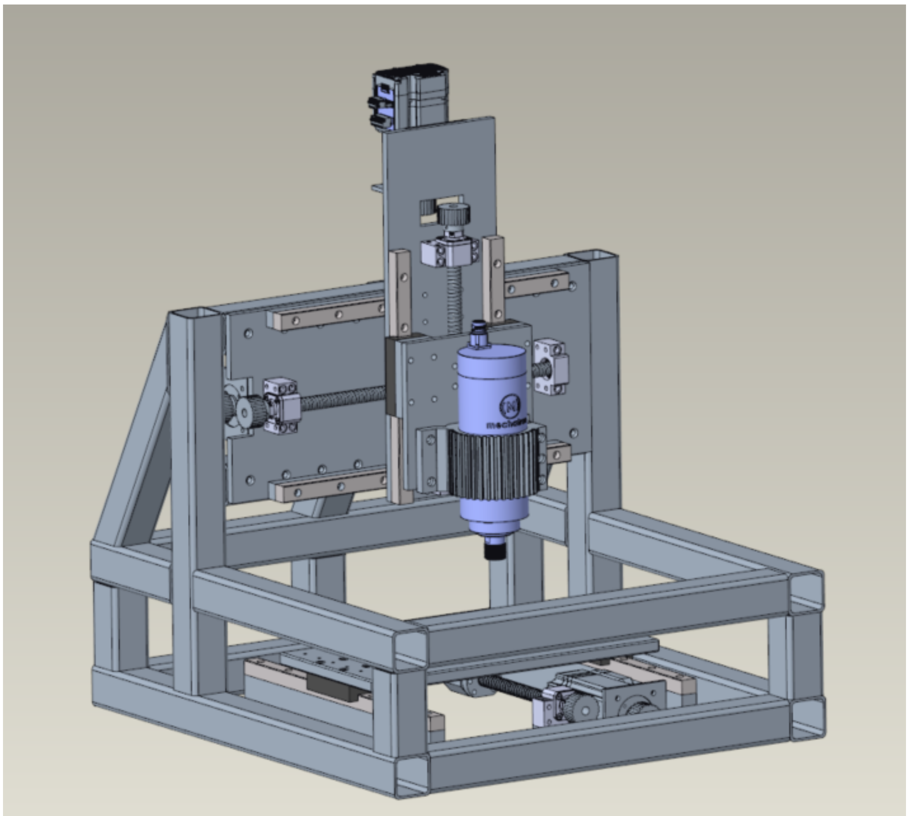
Designing a CNC Mill

Vilppu Jokinen

May 19th, 2025



Contents

1 Introduction: Desktop CNC Mill Project

In recent years, desktop 3D printers have revolutionized the world of prototyping, fabrication, and creative engineering. With the ability to transform an idea into a tangible plastic part in a matter of hours, additive manufacturing has become an essential tool for makers and engineers alike. One of its most compelling advantages is its design flexibility—complex geometries that challenge traditional manufacturing methods can be produced with minimal constraints on manufacturability.
However, despite the transformative potential of desktop 3D printers, they are not without their limitations. Most consumer-grade 3D printers are restricted to printing thermoplastics, such as PLA, PETG, ABS, and ASA. While these materials are well-suited to a wide range of applications, they quickly reach their limits in scenarios demanding higher stiffness, strength, or thermal resistance (e.g., operating temperatures exceeding 70°C). This restriction poses challenges for applications requiring robust and durable components.
To address these limitations, I set out to design and build a **desktop CNC mill** capable of machining a broader range of materials. The goal of this project is to create a compact yet powerful milling machine that can handle not only aluminum, wood, and acrylic but also mild steel—albeit in a carefully controlled and limited manner. By pushing the boundaries of what is possible with a home fabrication setup, this project seeks to bridge the gap between accessible 3D printing and more demanding subtractive manufacturing processes.

2 Use Case

In the introduction, I outline my motivation for designing a desktop CNC mill. As a passionate engineering student, I eagerly began creating initial drafts using CAD (Computer-Aided Design) software to visualize my concept for the machine. In this early phase, critical mechanical components—such as lead screws and linear rails—were selected based purely on guestimates. I quickly realized, however, that a structured approach grounded in calculations would not only be prudent but also essential for developing a robust design. This realization led me to document the design and analysis process comprehensively, resulting in the creation of this document.

Initially, I made the mistake of diving straight into calculations immediately following the introduction. However, it became clear that, without a clearly defined use case for the machine, I would be forced to make assumptions at nearly every step. This lack of a defined framework significantly increased the likelihood of errors and inconsistencies in the design. To address this issue, this chapter establishes a "worst-case" machining scenario for the mill.

By basing my calculations on this scenario, I can ensure that the machine will meet the demands of its most extreme operational conditions.

2.1 Description of the Use case

This desktop CNC mill is designed for hobbyists and engineering students to prototype small mechanical components while maintaining the capability to machine mild steel (e.g., S235). It will feature compact dimensions suitable for limited workshop space, achieve micrometer-level precision, and perform various machining operations such as contouring, pocketing, drilling, and light milling of steel. The mill will accommodate workpieces up to 300 mm x 300 mm and will be optimized to handle machining forces and spindle speeds typical for both aluminum and mild steel.

3 Machine Specifications and Practical Constraints

3.1 Size limitations

As said in the Introduction, the main goal of this design is to provide a complete, but compact solution in order to manufacture parts out of Aluminium. Ideally, the machine would be similar in size to a big desktop 3D-printer.

The machine should not exceed 700 mm in length, width or height.

3.2 Cutting Forces

The general equation used to calculate the cutting force goes as follows:

$$F_c = k_c * A * C_1 * C_2 \tag{1}$$

Here F_c is the cutting force in N, k_c is the specific cutting force in $\frac{\mathrm{N}}{\mathrm{mm}^2}$, A is the cutting area in mm^2, C_1 is a correction factor determined by the material of the cutting tool, C_2 is a correction factor given by the wear of the cutting tool.

k_c can be calculated with the following equation:

$$k_c = \frac{k_{c1.1}}{h^{m_c}} \tag{2}$$

$k_{c1.1}$ is a reference value for the specific cutting Force in $\frac{\mathrm{N}}{\mathrm{mm}^2}$, m_c is a constant given by the material.

A can be calculated as follows:

$$A = a_p * f_z \tag{3}$$

a_p is the depth of cut in mm and f_z is the feed per cutting blade in mm.

3.2.1 Cutting forces required for Aluminium

I would like to cut Aluminium with a depth of 2 mm and a height of 2 mm.
First we have to calculate k_c using the Equation (2), the values used for $k_{c1.1}$ and m_c were found in [1]:

$$k_c = \frac{k_{c1.1}}{h^{m_c}} \tag{4}$$

$$k_c = \frac{830}{0,23} \tag{5}$$

$$k_c = 3608,7 \frac{\mathrm{N}}{\mathrm{mm}^2} \tag{6}$$

Next, we have to calculate A using the equation (3), the value for a_p is an assumption made as a design constraint and f_z was determined with the help of [1]:

$$A = a_p * f_z \tag{7}$$
$$A = 2\text{mm} * 0,01\text{mm} \tag{8}$$
$$A = 0,02\text{mm}^2 \tag{9}$$

Assuming the use of a new High Speed Steel Endmill, we have following values for c_1 and C_2:

$$C_1 = 1,2$$

$$C_2 = 1,0$$

Combining all these Values and using them in Equation (1), gives us the following cutting force:

$$F_c = k_c * A * C_1 * C_2 \tag{10}$$
$$F_c = 3608,7\frac{\text{N}}{\text{mm}^2} * 0,02\text{mm}^2 * 1,2 * 1,0 \tag{11}$$
$$F_c = 86,6\text{N} \tag{12}$$

Now making the same calculations for a bigger milling operation. Using a Endmill with a diameter of 12 mm, we have $f_z = 0,049$, $a_p = 12mm$. Using these values, the calculations go as follows:

$$F_c = 3608,7 * 1 * 1,2 * 12 * 0,049 \tag{13}$$
$$F_c = 424,4\text{N} \tag{14}$$

3.2.2 Cutting forces required for Steel

When cutting steel we will limit to a depth of cut of 2 mm, as the mill should mainly be used to machine Aluminium or softer materials and only in some special cases mild Steel (S235 and similar).
Calculating k_c:

$$k_c = \frac{k_{c1.1}}{h^{m_c}} \tag{15}$$
$$k_c = \frac{1780}{0,17} \tag{16}$$
$$k_c = 10470,6\frac{\text{N}}{\text{mm}^2} \tag{17}$$

Calculating A:

$$A = a_p * f_z \tag{18}$$
$$A = 2\text{mm} * 0,009\text{mm} \tag{19}$$
$$A = 0,018\text{mm}^2 \tag{20}$$

And finally Calculating F_c:

$$F_c = k_c * A * C_1 * C_2 \tag{21}$$
$$F_c = 10470,6\frac{\mathrm{N}}{\mathrm{mm}^2} * 0,018\mathrm{mm}^2 * 1,2 * 1,0 \tag{22}$$
$$F_c = 226\mathrm{N} \tag{23}$$

Looking at the cutting Force values that come out for the various different milling operations, I will be setting the biggest allowed cutting force for this machine at $F_{c_{max}} = 500\mathrm{N}$.

3.3 Cutting speeds and Feed rates

3.3.1 Cutting and spinning speeds required for Aluminium Machining

Given that we now know approxiately what kind of Forces need to be exceeded in order to have a successfull milling operation, we can start to Discuss about the cutting speeds and feed rates that need to be achieved throug this design.
After a quick google search, or by looking into [1], we can se that typical cutting speeds for aluminium:

$$V_c \in [100; 200]\frac{\mathrm{m}}{\mathrm{min}}$$

.
Assuming, that we use an 6 mm Endmill, we can use the following equation to calculate the required spinning speed.

$$n = \frac{V_c * 1000}{\pi * d} \tag{24}$$

With n being the spinning speed and d being the diameter of the Endmill.

Using the previously mentioned assumptions, we get $n = 10610\frac{1}{\mathrm{min}}$
So in order to properly be able, to machine aluminium, with smaller Endmills, the spindle of the Mill should at least be able to turn at $n = 20000\frac{1}{\mathrm{min}}$.
Next we have to calculate the required power for the spindle.

$$P = M\omega \tag{25}$$
$$P = F_{c_{max}} * \frac{d}{2} * \frac{\pi * n}{30} \tag{26}$$
$$P = 500 * 0,003 * 1050 \tag{27}$$
$$P = 1575\mathrm{W} \tag{28}$$

As an option I have been considering and would see as one of the best fits a 2,2kW, air-cooled spindle, without an ATC(Automatic tool changer) and able to reach 24000 RPM, in combination with a variable frequence drive. The exact model I have been looking at is the HFSAC-8022-4-ER20. We will go into more detail concerning the spindle selection in another section of this document.

3.3.2 Feed rates

In this section I will be calculating the required feed rates fr the mill, and make some rough calculations, in order to get a order of magnitude for the motors and ball screws that will be required for this machine.

$$V_f = n * f_z * z \tag{29}$$
$$V_f = 10610 \frac{\mathrm{U}}{\mathrm{min}} * 0,01\mathrm{mm} * 4 \tag{30}$$
$$V_f = 66,688 \frac{\mathrm{rad}}{\mathrm{min}} * 0,01\mathrm{mm} * 4 \tag{31}$$
$$V_f = 2667,3 \frac{\mathrm{mm}}{\mathrm{min}} \tag{32}$$
$$V_f = 2,67 \frac{\mathrm{m}}{\mathrm{min}} \tag{33}$$
$$V_f = 0,0445 \frac{\mathrm{m}}{\mathrm{s}} \tag{34}$$

According to these calculations, the axis of the machie should be able to provide a feed rate $V_F = 2,67 \frac{\mathrm{m}}{\mathrm{min}}$, all while overcoming the cutting force of 500 N. Using a sizing tool made by the Company HIWIN, which specializes in the design and production of linear motion systems, I will make the assumption, that a ball screw with a outer diameter of 16 mm and a pitch of 5 mm should work for this application.

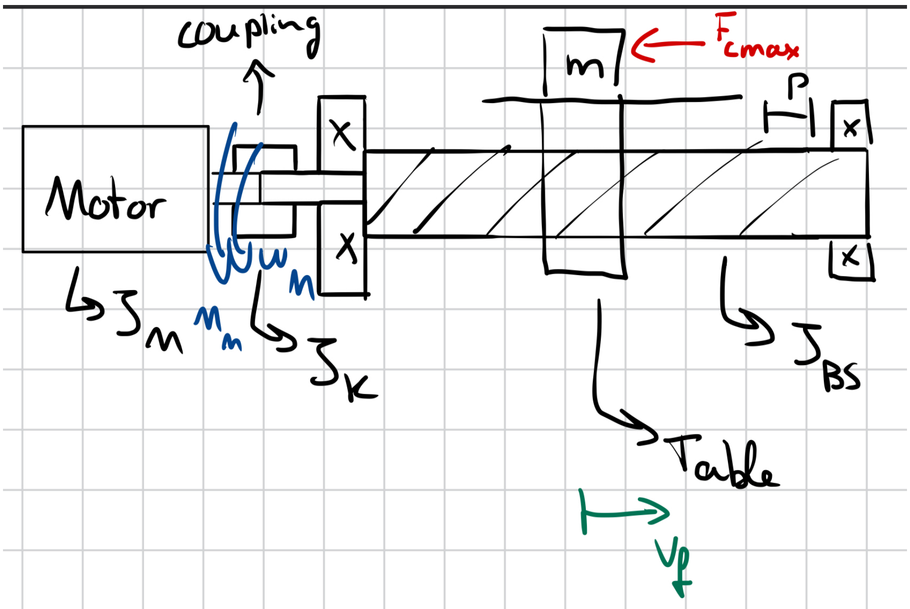


Figure 1: Simplified sketch of the power trainsystem of an axis

Figure 1 goves us the variables that will be determining for the calculation of the axis. We will begin by calculating the Couple that the motor has to endure in a stationary use.

$$M_m = \frac{F_{cmax} * p}{2 * \pi} \tag{35}$$
$$M_m = \frac{500\text{N} * 0,005\text{m}}{2\pi} \tag{36}$$
$$M_m = 0,398\text{Nm} \tag{37}$$

Assuming the spindle has a efficiency of $\eta = 0,85$, which is a standard value for the efficiency of ball screw spindles, we get:

$$M_m^* = \frac{M_m}{\eta} \tag{38}$$
$$M_m^* = \frac{0398}{0,85} \tag{39}$$
$$M_m^* = 0,468\text{Nm} \tag{40}$$

Calculating the total moment of inertia of the power train, we get:

$$J_{tot} = J_m + J_k + J_{BS} + \frac{mv_f^2}{\omega_m} \tag{41}$$
$$J_{tot} = (0,000068 + 0,00005 + 0,0000252 + 0,0000127)\text{kgm}^2 \tag{42}$$
$$J_{tot} = 0,00016\text{kgm}^2 \tag{43}$$

These values will be used in a further section to analyze the dynamics of the power train. But next we will calculate the required Power:

$$P_m = M_m^* \cdot \omega_m \tag{44}$$
$$P_m = 0,468\text{Nm} * 57,2\frac{\text{rad}}{\text{s}} \tag{45}$$
$$P_m = 26,8\text{W} \tag{46}$$

As we can see the required Mechanical Power is pretty low for the stationary state. The need for higher power motors comes not from the stationary use of the machine, but of the required accelertions in order to assure fast milling operations.

4 Designing the frame

4.1 Introduction to the design of the frame

The structural integrity of a machine tool's frame is paramount to its operational efficacy and precision. A robust and rigid frame is indispensable for maintaining the quality of produced components, ensuring they meet stringent tolerance specifications. During machining processes, the frame must effectively endure and counteract the forces exerted, while ideally damping vibrations to achieve superior surface finishes and dimensional accuracy. Without such structural stability, a machine tool cannot consistently deliver parts of high quality, underscoring the critical role of the frame in the machine's overall performance.

The machine frame not only serves as a supporting structure but also acts as a platform for mounting additional essential components. This includes the installation of guide rails, drive systems, and other mechanical elements necessary for the machine's operation. A precisely engineered frame enables optimal integration of these components, ensuring their seamless functionality. Furthermore, the frame should be designed to allow for extensions and modifications to meet the specific requirements of various machining processes. This contributes to maximizing the machine's versatility and adaptability.

4.2 Frame type

As I began conceptualizing the frame design, the initial decision revolved around selecting the appropriate frame type for my machine. I explored various options, including knee frames, L-shaped frames, and portal frames. Ultimately, I opted for a hybrid design that combines the characteristics of both the L-shaped and portal frames. This approach offers an efficient use of space and simplifies the fabrication process compared to other frame types. Such a combination leverages the strengths of each frame style, ensuring a balance between structural integrity and practicality.

In this subsection, I will address the choice of profiles and beams for the frame construction. Initially, I considered using aluminum extrusion profiles, such as Rexroth Profiles, due to their ease of fabrication and assembly, as well as the availability of off-the-shelf connectors. However, aluminum lacks the stiffness of steel, and the connectors can be less rigid compared to welded joints. Consequently, I decided to construct the frame using welded steel tubing, which offers superior structural integrity. Additionally, this approach allows for the strategic incorporation of sheet metal in areas where enhanced structural support is necessary. For information the steel tubing is 50 mm * 50 mm, and has a wall thickness of 3 mm.

The design I eventually came up, after making a lot of improvements, thanks to the FE-Analysis is shown in Figure 2.

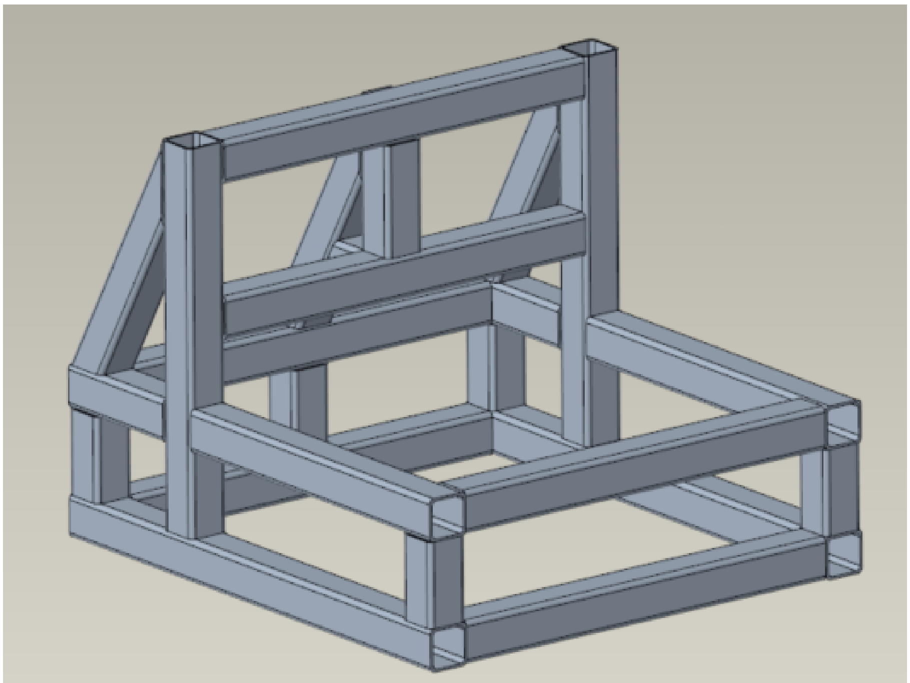


Figure 2: 3D-Model of the Frame of the machine

4.3 FEM-Simulation

To verify the design I created using CAD, I conducted a Finite Element Analysis of the frame. In this subsection, I will present the model I developed, describe the setup of the analysis, and critically evaluate the results of my work.

4.3.1 Fundamentals and Model setup

One of the most important aspects of conducting a Finite Element Analysis (FEA) is the model setup. This step is crucial, as the model must closely reflect reality to provide accurate insights into the frame's deformation and stress scenarios during various machining operations.
The first step in setting up the FE model is importing the geometry into the FEA software. For the following simulations, I used the student version of Ansys Workbench 2025 R1 [2]. The 3D model was initially created using Creo Parametric 11 [3], and to utilize it within Ansys Workbench, I exported it as a STEP file before importing it into the software.
The model used for the FEA is shown in Figure 3. It primarily consists of the frame and the ground plate for the machine's X-axis. Notably, certain features, such as drill holes, were omitted from the frame model to facilitate smoother meshing and improve mesh quality.

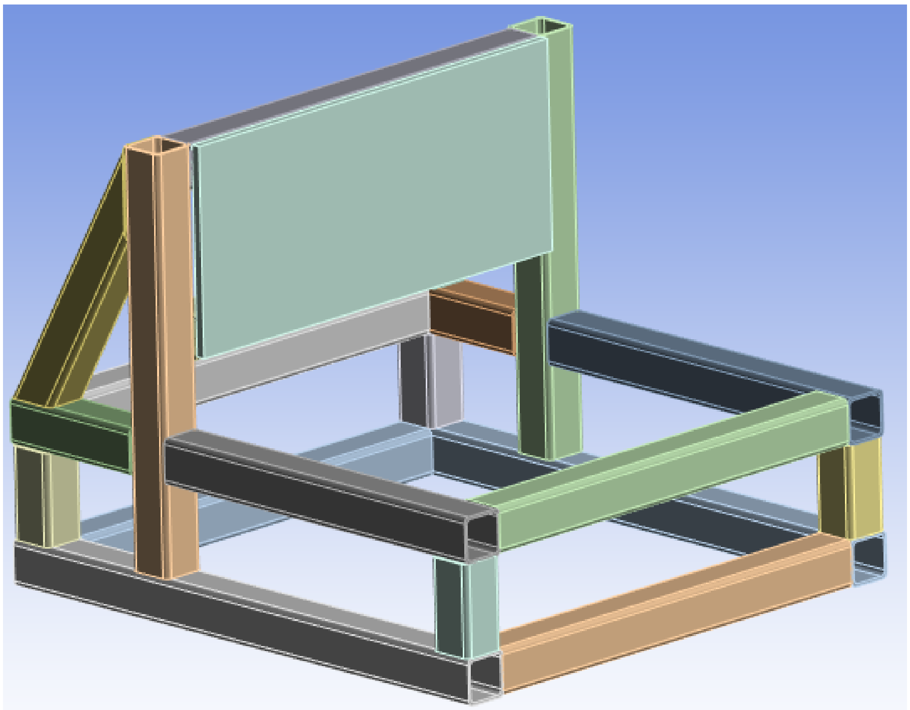


Figure 3: 3D-Model used for the FEA

After importing the model into the simulation software, I defined the material characteristics required for the analysis. Since the frame will be made of steel, I used the following material parameters in the simulation:

$$E = 210\text{GPa}, \mu = 0,3, \rho = 7850\frac{\text{kg}}{\text{m}^3}$$

As mentioned above, the frame is constructed using steel tubing, which is joined through welded connections. To accurately represent this connection type in the model, all contact regions were defined as "Bonded," as this contact type provides the most precise simulation results for welded joints.
Regarding the boundary conditions, since the machine will be mounted on a table or other workbench surface, I chose to apply a fixed support to the entire underside of the frame for this simulation. The area where this constraint was applied is shown in Figure 4.

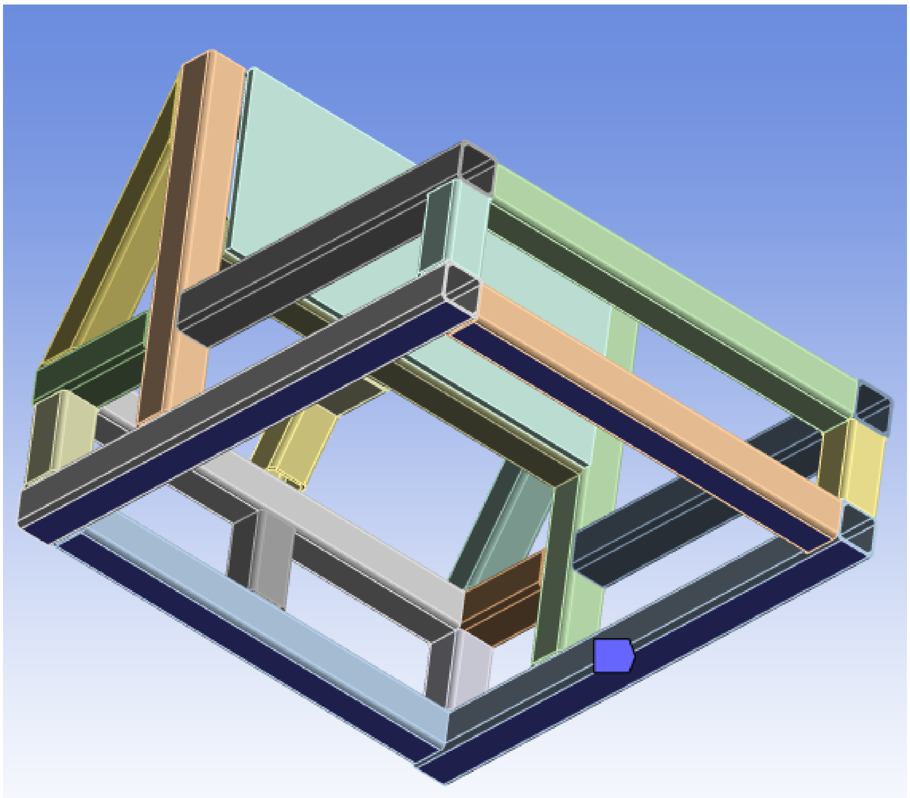


Figure 4: Bottom Area where the fixed support constraint was placed

As for the loads that the machine will have to endure, I decided to simulate the results for the maximal milling operation in the direction of one axis (for simplicity, I will just have to keep in mind not to exceed this limit). After some calculations I figured out the Force and moment that will be applied on to the groundplate of the X-Axis. The applied force is described in Figure 5 and 6, where as the resulting moment is described in figures 7 and 8.

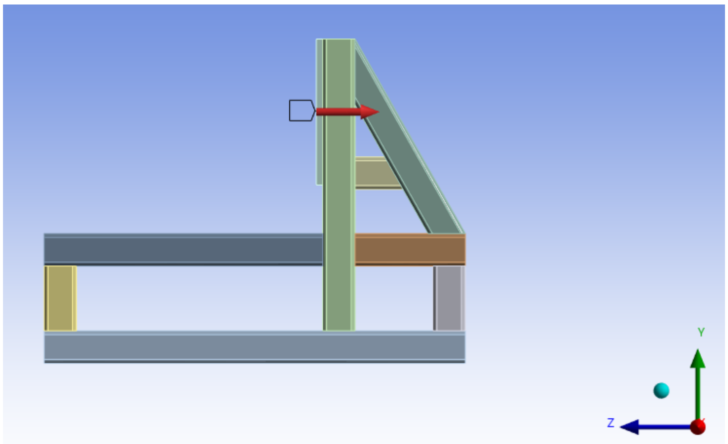
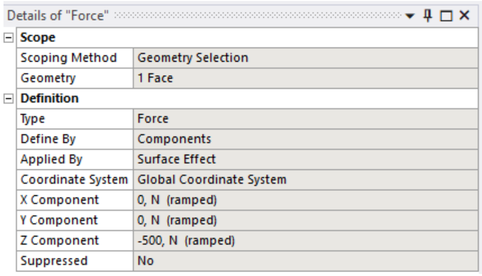


Figure 5: Geometrical representation of the Force acting on the frame



Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Force
Define By	Components
Applied By	Surface Effect
Coordinate System	Global Coordinate System
X Component	0, N (ramped)
Y Component	0, N (ramped)
Z Component	-500, N (ramped)
Suppressed	No

Figure 6: The specifics about the applied force

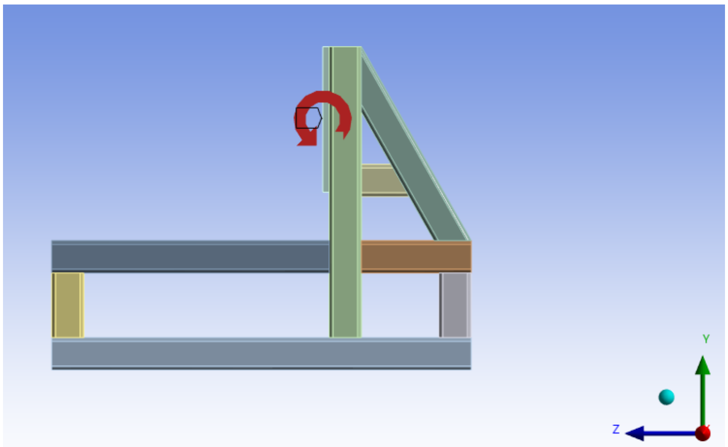
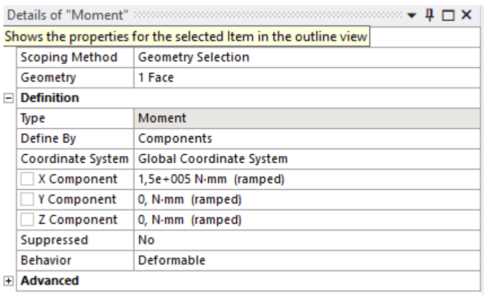


Figure 7: Geometrical representation of the Moment acting on the frame



Details of "Moment"	
Shows the properties for the selected Item in the outline view	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Moment
Define By	Components
Coordinate System	Global Coordinate System
X Component	1,5e+005 N·mm (ramped)
Y Component	0, N·mm (ramped)
Z Component	0, N·mm (ramped)
Suppressed	No
Behavior	Deformable
Advanced	

Figure 8: The specifics about applied Moment

Another crucial model parameter in Finite Element Analysis (FEA) is meshing, as an FE analysis would not be possible without it. To begin, the best practice is to start with a relatively large element size to gain an initial understanding of the expected stress and deformation values. Then, simulations should be run with progressively smaller elements until the results no longer change significantly. At this point, the model is converging toward a result, which is typically a reliable approximation of real-world behavior.
If the model fails to converge, the issue is known as a singularity. Singularities do not exist in reality but appear in FE simulation results due to the iterative mathematical processes behind the visualization in FEM software.
Having explained best practices, I will now describe my approach. Due to the limitations of the student version of Ansys, such as constraints on element size, I was unable to conduct a full

convergence analysis. Instead, I ran the simulation using the smallest element size available, while ensuring a finer mesh for critical beams and a coarser mesh for beams farther from the load. The mesh obtained under these constraints is shown in Figure 9. The smallest elements are 6 mm, the largest are 80 mm, and intermediate elements measure 15 mm. While I cannot be certain that this mesh is sufficiently fine, I hope it provides an adequate representation

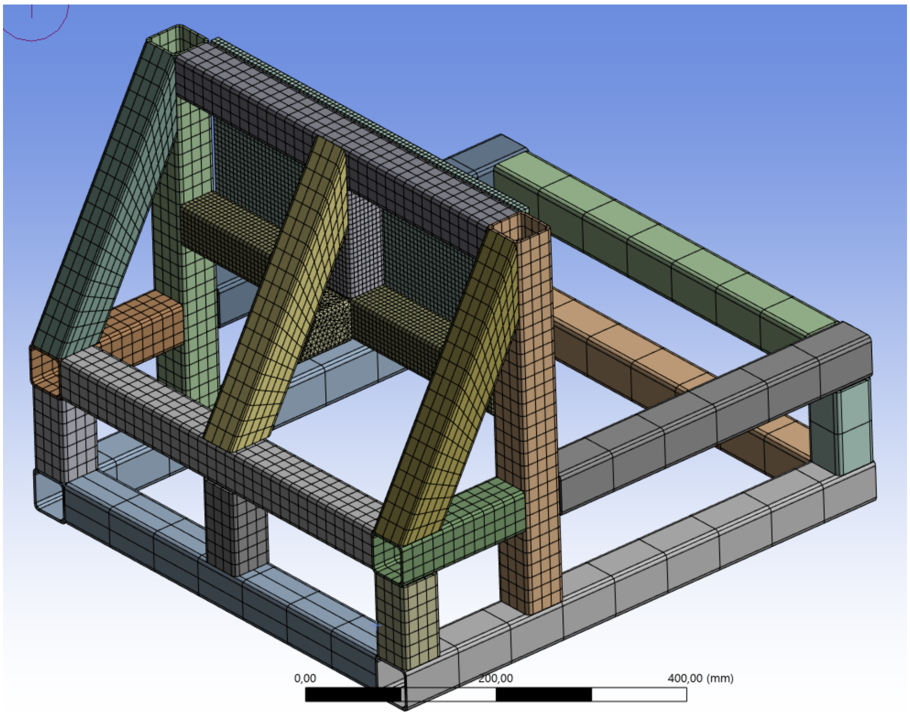


Figure 9: Meshing of the Frame for the FEM-Simulation

4.3.2 Results of the FE-Simulation

In this section of the documentation, we will analyze the simulation results. The primary objective of the simulation is to estimate the deformation of the frame during milling operations, ensuring that the design meets the predefined performance criteria. Additionally, I will briefly examine the stress distribution within the frame to verify that no beam is overloaded or at risk of plastic deformation.
First, we will evaluate the deformation occurring during the milling operation. The simulation results are presented in Figure 10.

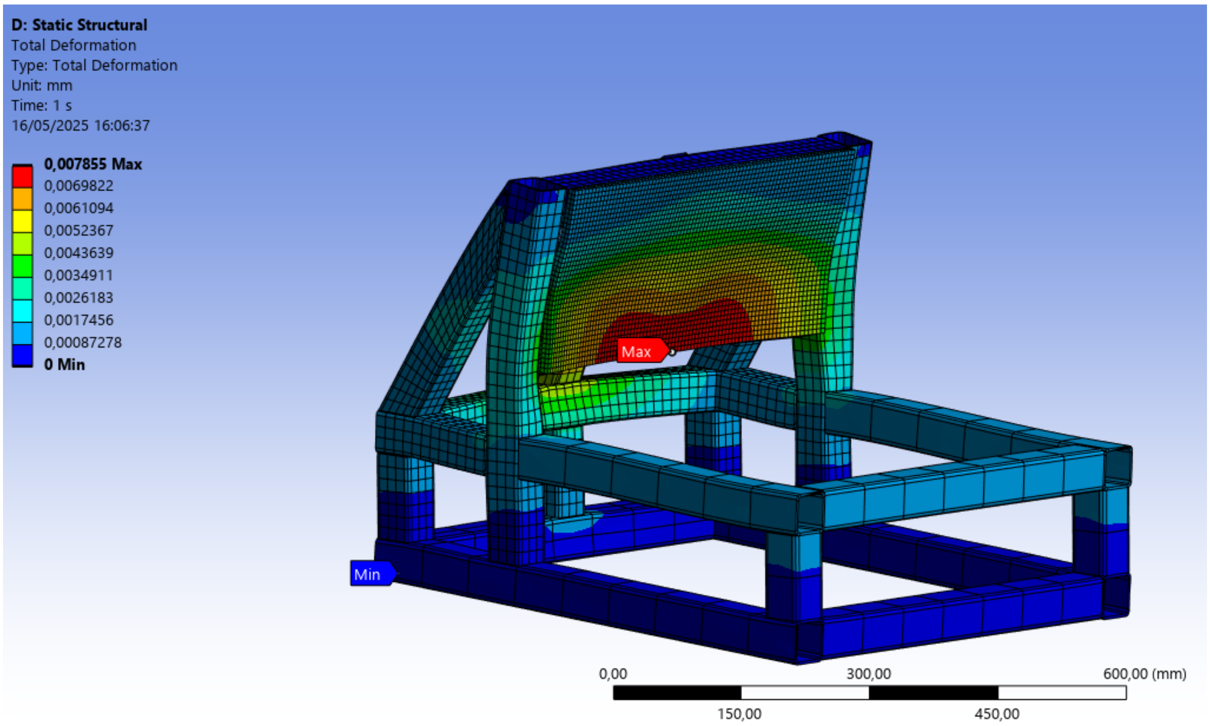


Figure 10: Simulation results for the deformation of the frame

The simulation results indicate that the greatest deformation occurs at the center of the base-plate of the X-axis, which aligns with expectations. The total deformation is 0.007855 mm, remaining below the predefined target of 0.01 mm for this machine.
It is important to emphasize that these results represent a worst-case scenario, meaning the deformation should be significantly lower during standard operations. Additionally, the linear rails, once installed, may further enhance the structural stiffness.
Considering these factors, this simulation provides a conservative assessment of the frame's deformation, suggesting that the design remains within acceptable limits.
Let's take a look at the stress state of the frame, shown in figure 11.

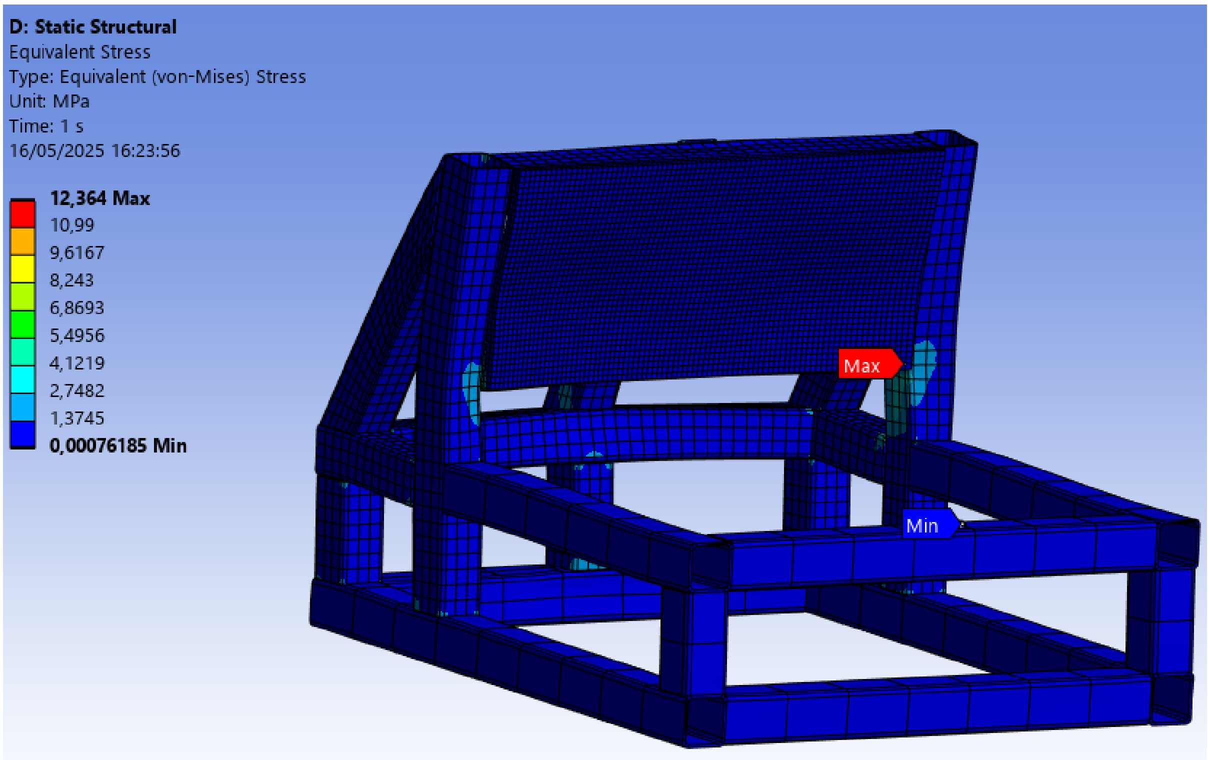


Figure 11: Simulation results for the stress state of the frame

As we can see, the highest stress that occurs in the Frame is approximately 12,4 MPa. This is far underneath the yield strength of the Material I plan to use ($Re_{S235JR} = 235\text{MPa}$). Therefore the Frame shouldn't be overloaded and it should be able to easily withstand the loads of the milling operations.

4.3.3 Critical review of FE-Simulation

Based on the results of the simulation, the frame I designed should be capable of withstanding the milling operations without exceeding the allowable deformation limits.
While simulation is an invaluable tool for predicting structural behavior, it is important to acknowledge its limitations. Real-world conditions often deviate from simulated scenarios, despite continuous advancements in simulation technology. Several factors can introduce uncertainties or lead to discrepancies between simulation results and actual machine performance.
For this particular machine, potential sources of error include inaccuracies in calculating the milling force and the resulting moment, as well as variations in how these forces are transmitted into the frame. Additionally, simplifications made during model preparation, such as removing small features for meshing efficiency, may slightly affect the accuracy of the results.
Another limiting factor is the restricted capabilities of the student version of Ansys Workbench, which imposes constraints on element size and convergence analysis. This limitation may affect the precision of stress and deformation predictions, preventing a fully refined simulation. Furthermore, assumptions regarding material properties—such as uniformity in mechanical characteristics—might not perfectly align with real-world manufacturing tolerances or potential imperfections in welds or connections.
Environmental conditions, such as temperature fluctuations, vibrations, and external forces, can also influence the actual performance of the frame in ways that are difficult to account for in a purely computational model. Finally, modeling constraints related to boundary conditions

and load applications may differ slightly from the real-world setup, affecting stress distribution and deflection predictions.
Considering these factors, while the simulation provides a strong indication that the design meets performance requirements, experimental validation—such as physical testing or additional real-world observations—remains a crucial step in ensuring the reliability of the frame under operational conditions.

5 Drive Systems

6 The Controls and Electronics

7 Conclusion

This project is still ongoing and isn't finished yet. Therefore, I am eager to hear about any improvements I should/can make to the design or about mistakes I might've made in the design process. Feel free to contact me via Github or any other communication channel you can find on my page.

8 Sources

[1]: Tabellenbuch Metall, Europa Lehrmittel, 47.Auflage 2017
[2]: https://www.ansys.com/de-de/academic/students , Consulted on: 14.05.2025
[3]: https://www.ptc.com/en, Consulted on: 14.05.2025