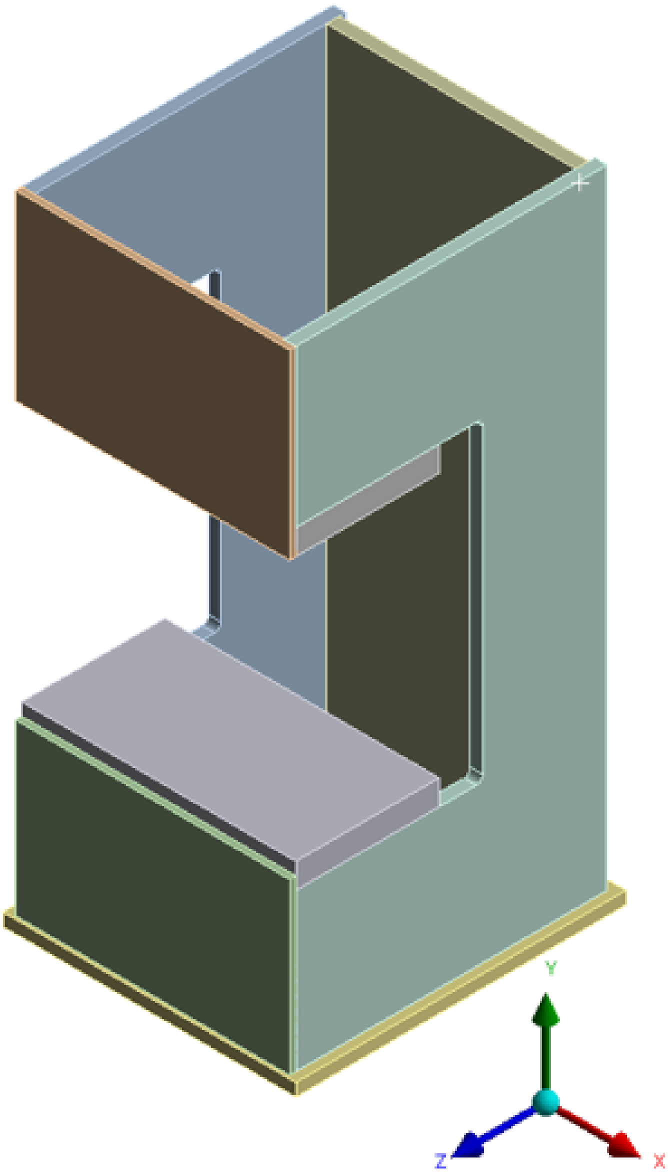
FE-Analysis of a Press Frame

Paul Idt, Vilppu Jokinen, Johannes Xue

January 2025



Contents

1 Introduction

This is the documentation for the structural analysis of a C-frame press.

The structural analysis examines the stiffness and strength of the press frame. Regarding stiffness, special attention was paid to the deformation of the lower plate and the opening angle of the frame. In terms of strength, the equivalent stresses occurring in the frame under load were analyzed, with a particular focus on the fillets located on the side plates of the welded frame.

In order to meet the given design requirements, some modifications were presented and proofed. These modification include a bigger radius for the fillets and a thicker lower plate.

PS: This Group Project was done for a module called "Computer Aided Engineering". The idea of this Project was, for the students to conduct an FE-simulation, analyse the results and to produce a good documentation, following the various steps of the project.

2 List of Documents

The Project was done using the following simulation softwares:

1. ANSYS 2020 R2
2. ANSYS Workbench 2020 R2
3. ANSYS Mechanical 2020 R2

CAD-files, the task description and guidelines for the documentation were given to the students in the following moodle-course:

4. Moodle-Course "4 Computer Aided Engineering - FEM(Block)" (09.01.2025)

Material properties such as Young's modulus or Poissons ratio were taken from:

5. Technisches Taschenbuch, Schaeffler Technologies AG & Co. KG, 2017

3 Task Description

The design department of a company has developed a new C-frame press, which is shown in the Figure 1.

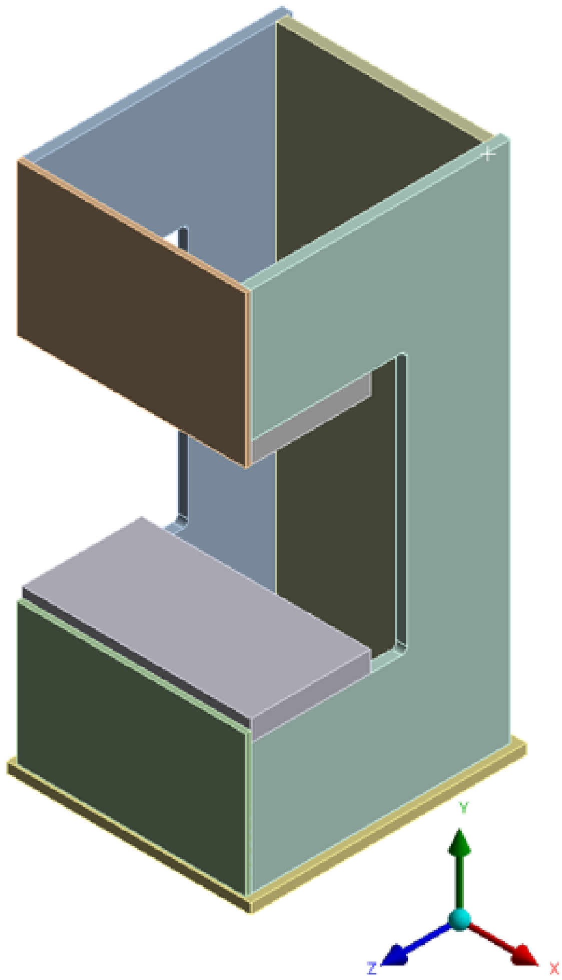


Figure 1: CAD-Model of the Press Frame

A finite element (FE) analysis is to be conducted to optimize the stiffness of the structure. The focus of the analysis is on the opening angle of the frame under load and the deformation of the lower plate.

For strength assessment, the stresses in the frame must be analyzed, with particular attention to the fillets on the side plates, as they represent stress concentration points.

Finally, design modifications must be implemented to ensure that the maximum deflection of the lower plate under load is limited to 0.05 mm.

Regarding the frame, the design modifications must not result in an equivalent stress exceeding 50 MPa.

4 Fundamentals and Model Setup

The supposed material of the press frame, is 1.0038, in Germany better known as S235JR, one, if not the most used steel. It has a Young's modulus $E = 210\,\text{GPa}$ and a Poisson's ratio of $\mu = 0.3$.

The different parts of the frame are welded together, therefore the contact type chosen for the frame was set to "Bonded", as shown in the figures 2 and 3.

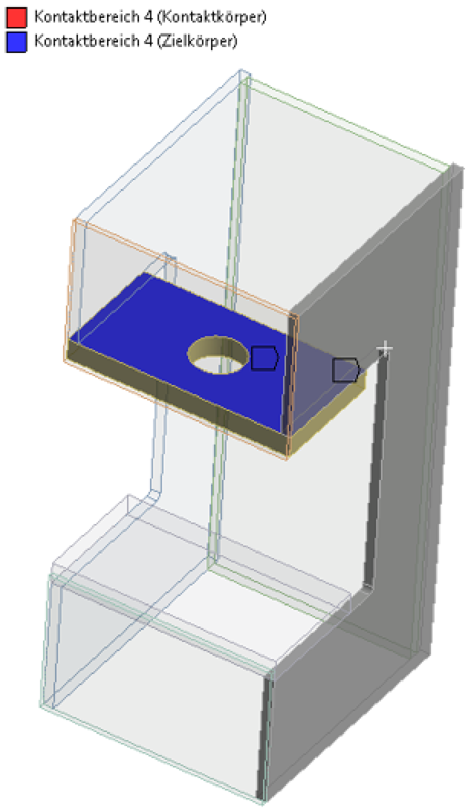


Figure 2: Visualization of a single Contact Area

Details von "Kontaktbereich 4"

Zuweisung	
Zuweisungsmethode	Geometrieauswahl
Kontakt	1 Fläche
Ziel	1 Fläche
Kontaktkörper	Component1\Part 5
Zielkörper	Component7\Part 1
Geschützt	Nein
Definition	
Typ	Verbund
Kontaktfindung	Automatisch
Verhalten	Programmgesteuert
Kontaktfläche trimmen	Programmgesteuert
Trim Toleranz	6,0285 mm
Unterdrückt	Nein

Figure 3: Details of the contact arrea from Figure 2

In order to minimize the computation time, the base plate was removed as it doesn't improve the rigidity of the frame. As support Boundary, a fixed support was selected and this was set on the blue area shown in figure 4.

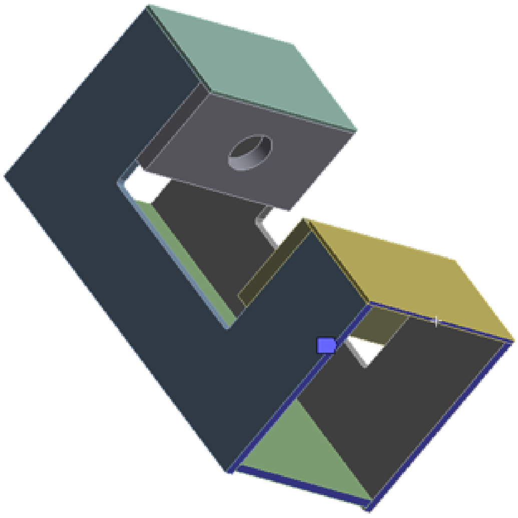


Figure 4: Support Area of the model

The press frame is set under load through two forces, with a magnitude of $F = 100\,\text{kN}$. These have opposite directions and the places where these are induced are shown in the figure 5.

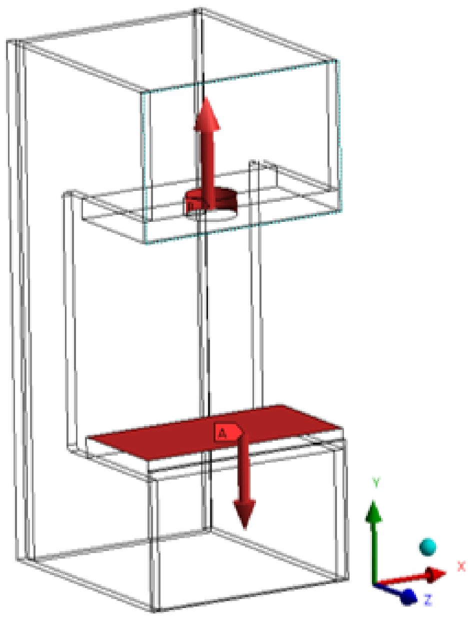


Figure 5: Areas where the forces are induced in to the press frame

5 Mesh Setup

Following a convergence analysis for the highest stress in the fillets of the frame and the biggest deformation in the lower plate, the following mesh parameters were used in this FEA. The overall mesh-size was set to $e = 60\,\mathrm{mm}$ and the used meshing method is the automatic one, as shown in figure 6. For the lower plate the element size was reduced to $e = 8\,\mathrm{mm}$ and for the fillet zones the element size was set to $e = 1\,\mathrm{mm}$, as shown in figrue 7 and 8.

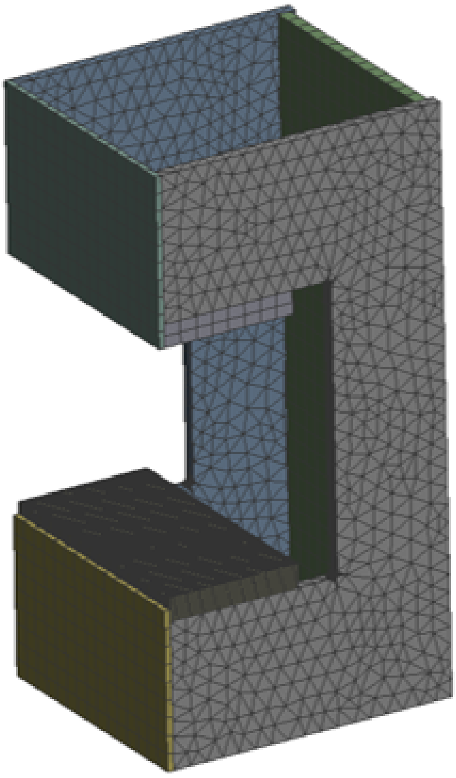


Figure 6: Overall mesh

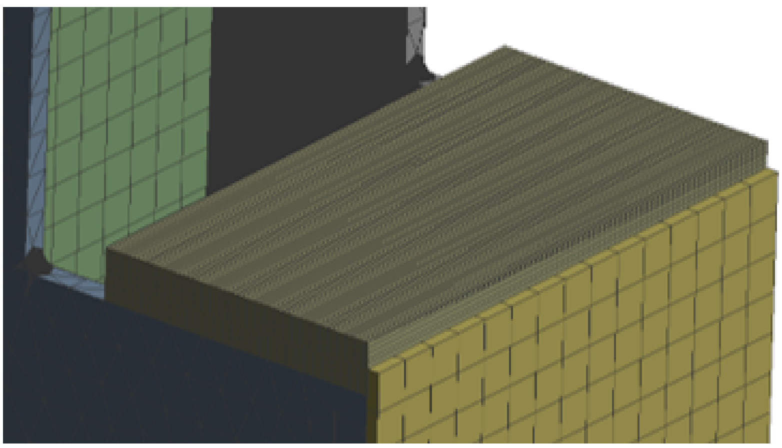


Figure 7: Mesh of the lower plate

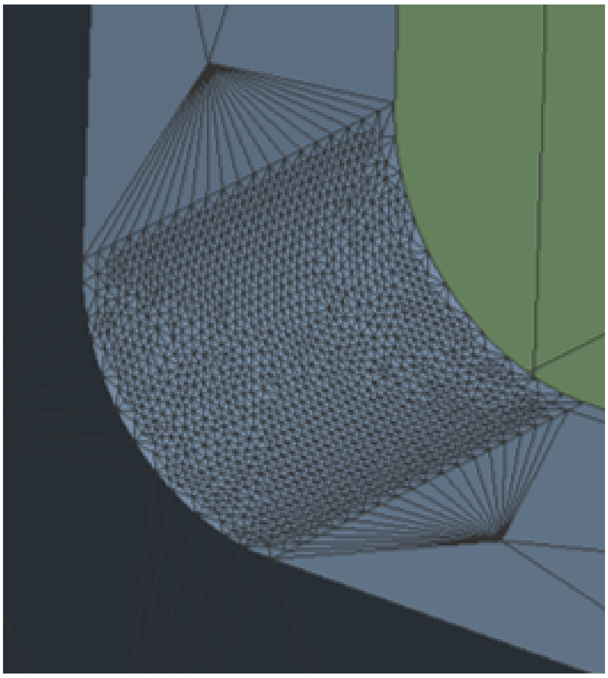


Figure 8: Mesh in the fillets of the frame

6 FEA Documentation

6.1 Equivalent Stresses

Unsurprisingly, the highest equivalent stresses are found in the fillets of the press frame, because these induce a stress concentration. These zones are critical for the design, and they are the limiting factor for the strength of the frame. The results of the FEA can be seen in Figure 9 and 10.

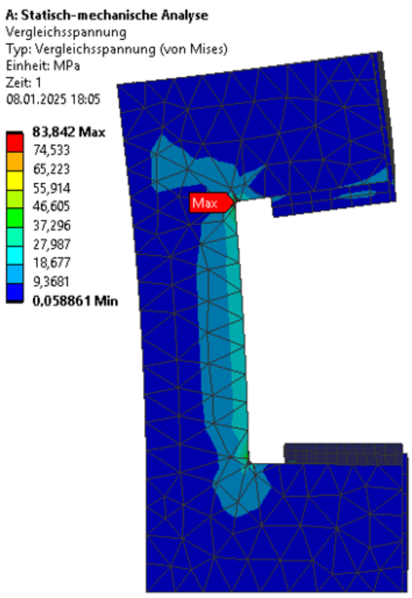


Figure 9: Equivalent stresses overall

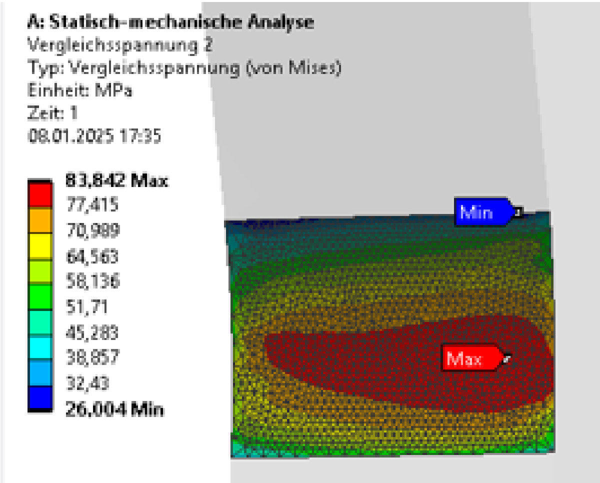


Figure 10: Equivalent stresses in the fillet

The maximum value of the equivalent stresses $\sigma_v = 83.8\,\text{MPa}$, is well above the required 50 MPa. A possible design modification, which would increase the strength of the frame, is to make the radius of the fillets larger.

By increasing the radius of the fillets from 20 mm to 75 mm, the maximum equivalent stress σ_v goes from 83.8 MPa to 48.9 MPa, as shown in figure 12.

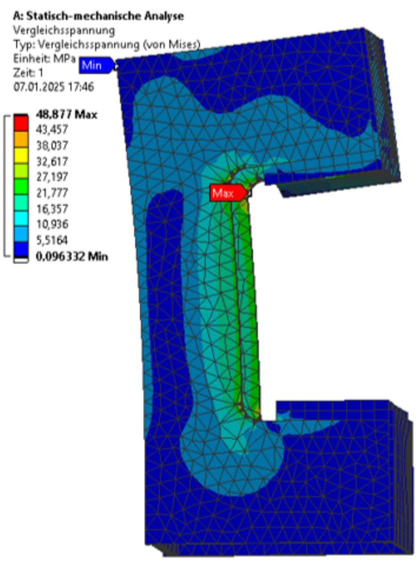


Figure 11: Equivalent stresses in the fillet after increasing fillet radius

Thanks to the increase of the radius from 20 mm to 75 mm, the maximum equivalent stress is now below the allowed 50 MPa.

6.2 Deformation of the lower plate

The stiffness of the lower plate was another design constraint given in the task description. In order to make a statement about the stiffness of the lower plate, the overall deformation was analysed as shown in figure 13.

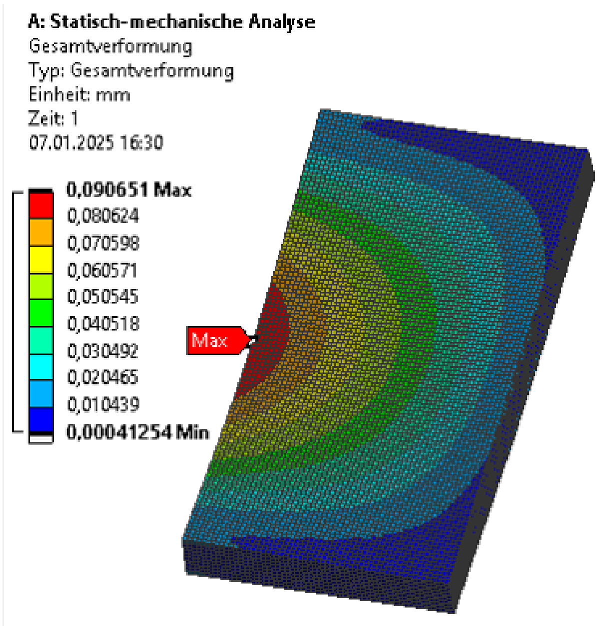


Figure 12: Deformation of the lower plate

Figue 13 shows that the maximal deformation is about 0.091 mm. This is higher than the allowed deformation of 0.05 mm, therefore some design changes must be applied. A possibility to increase the stiffness of the lower plate would be to make this thicker or you could try to add some kind of rod to support it from underneath.

By increasing the thickness of the lower plate from 80 mm to 110 mm, the maximum deformation from the lower plate sinks to 0.044 m, as shown in figure 14.

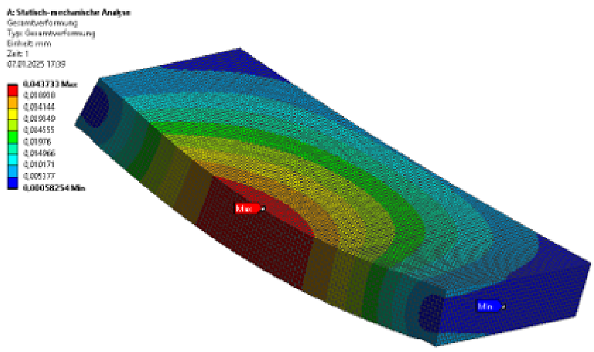


Figure 13: Deformation of the lower plate

Due to the increase in thickness the maximum deformation is now below the allowed 0.05 mm.

6.3 Opening angle of the frame

In order to determine the opening angle of the steel frame, the deformations in the y-direction of the upper plate are observed, as shown in Figure 14.

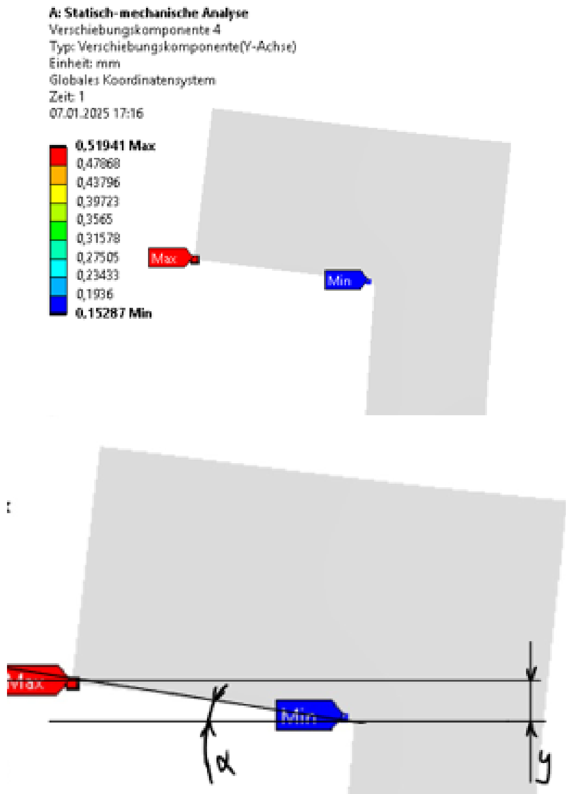


Figure 14: Opening angle of the press frame

After this, the following formula is used to calculate the opening angle.

$$\alpha = \arctan(\frac{y}{L}) \tag{1}$$

Where α is the opening angle, y the difference in deformation in the y-direction of the maximum and minimum of the upper plate and where L is the distance between the two measuring points.

With $L = 580\,\text{mm}$, $\alpha = 0.037$

7 Summary

Based on the conducted FE analysis, it was determined that the original design from the engineering department did not meet the requirements, making design modifications necessary. By adjusting the design and verifying it through an additional FE analysis, the requirements for the stiffness of the lower plate and the strength of the steel frame were met. For this purpose, the thickness of the lower plate was increased from 80 mm to 110 mm, and the fillet radii of the side plates were enlarged from R20 to R75.