KOÇ UNIVERSITY
COLLEGE OF ENGINEERING

MECH 303

Machine Design

Project 2: Computational Investigation on the Safety Factor of a Pin Bracket under Different Loading Scenarios

Name: Hakancan Öztürk

Submission Date: 26 December 2022

Objective

The safety factor of a pin bracket, which is a type of structural component used in various engineering applications, is an important consideration in the design process. It determines the margin of safety that the bracket can withstand before experiencing failure or damage. In this computational investigation, we will examine the safety factor of a pin bracket under different loading scenarios using the Static Structural module of ANSYS. By analyzing the behavior of the bracket under various loading conditions, we can better understand its performance and identify any potential weaknesses or critical regions within the geometry. This information can be used to optimize the design of the pin bracket and ensure its safe and effective operation.

Methodology

The importance of safety factors from an engineering point of view was explained in the objective part. In this experiment, we are investigating the different loading scenarios and for all of them, separate Static Structural analysis must be performed. There are three main steps to performing this analysis to ultimately compute the safety factors.

The first step is to set up the study. We created a new project in ANSYS and selected the Static Structural on the Workbench since the study we are conducting focuses on static loading on a 3d structure. We then, modeled the two different parts of the main structure namely, the bracket and the pin using the built-in CAD software, SpaceClaim. The bracket geometry and the pin were modeled according to the technical draft provided with the project instructions. A critical nuance in this part was to create another body within the CAD software so that the pin and the bracket are not treated as a single body. The 3d model can be seen in Figure 1.

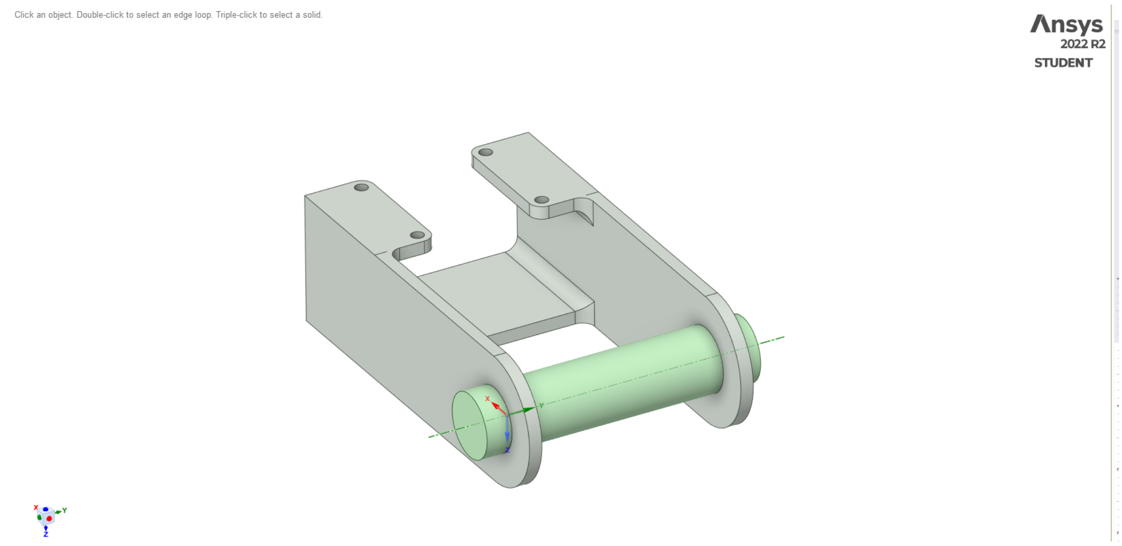


Figure 1. The modeled problem geometry in SpaceClaim

After designing the geometry, the material properties of the structure were adjusted according to the problem statement. The study specifies that the material is structural steel with a yield strength of 250 MPa. New material can be defined in the Engineering Data tab or the pre-existing material can be adjusted.

In the Model tab, the meshing was tuned as much as the Student Edition of ANSYS allows, which will be elaborated on in the discussion part. For the meshing, we followed different paths for the different parts of the system. For the bracket, the mesh elements were specified as tetragonal, and the mesh element sizes were specified as 6.0 mm. Additionally, the pin meshing was tuned by using edge sizing with the specification of 50 divisions around the edge. The results of the meshing can be seen in Figure 2.

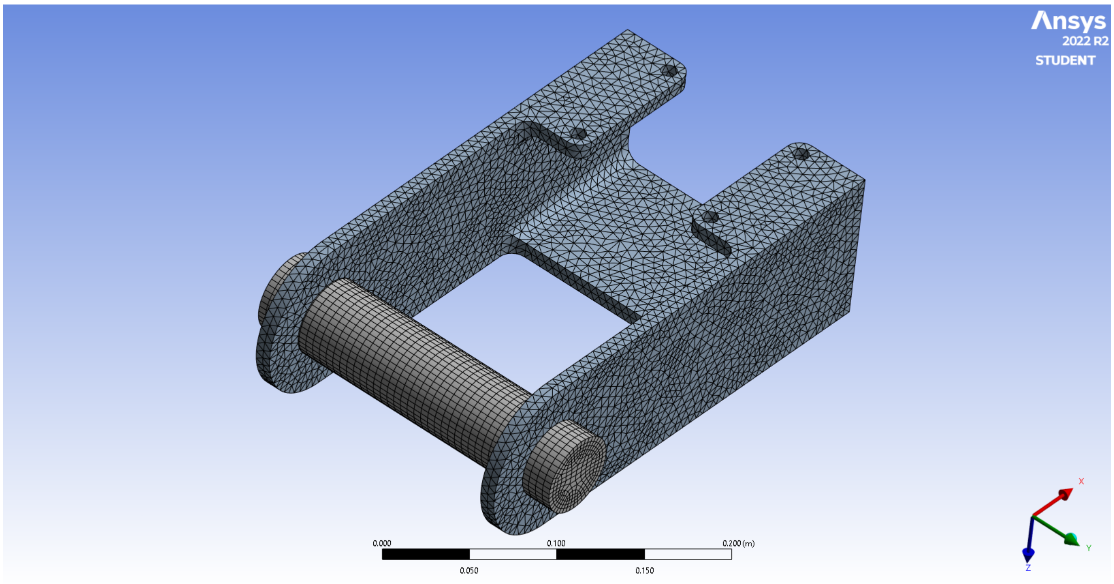
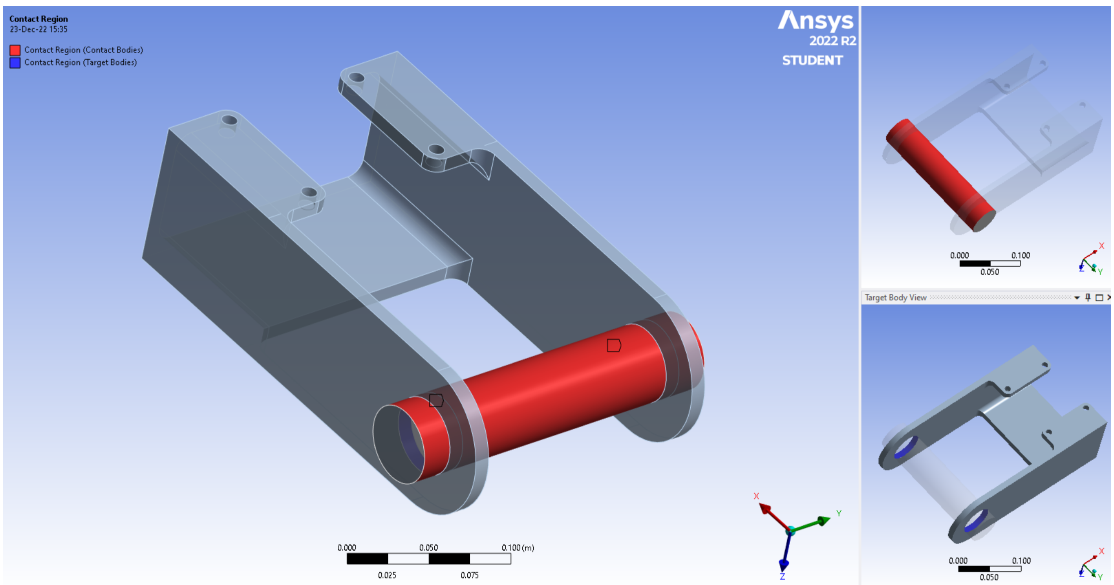


Figure 2. The meshing of the problem geometry

The behavior between the two bodies was also introduced in this step. Since we separated the bodies in the modeling part, the relationship between the bodies was set as frictional with k= 0.2, as can be seen in Figure 3.

a-



b-

Details of "Frictional - pin\Solid To SYS\S

Scope	
Scoping Method	Geometry Selection
Contact	1 Body
Target	2 Faces
Contact Bodies	pin\Solid
Target Bodies	SYS\Solid
Protected	No
Definition	
Type	Frictional
Friction Coefficient	0.2
Scope Mode	Manual
Behavior	Program Controlled
Trim Contact	Program Controlled
Suppressed	No
Display	
Advanced	
Geometric Modification	

Figure 3. The contract specifications between the pin and bracket, a) the geometry of the contact region, b) the properties of the contact

After the mesh tuning, the boundary conditions, and the initial conditions were specified. Since we will be changing the loading conditions for every case of the study the forces are added on the free surfaces of the pins and renamed for further ease. Then the first case loading was specified and the three components of each face were parametrized so that we can change them easily. The boundary conditions were very straightforward, the four pin holes on top of the bracket were set as fixed support, as can be seen in Figure 4. Note that the coordinate system given in the instructions differs from ours', therefore the given tabulated forces were altered to yield the same loading.

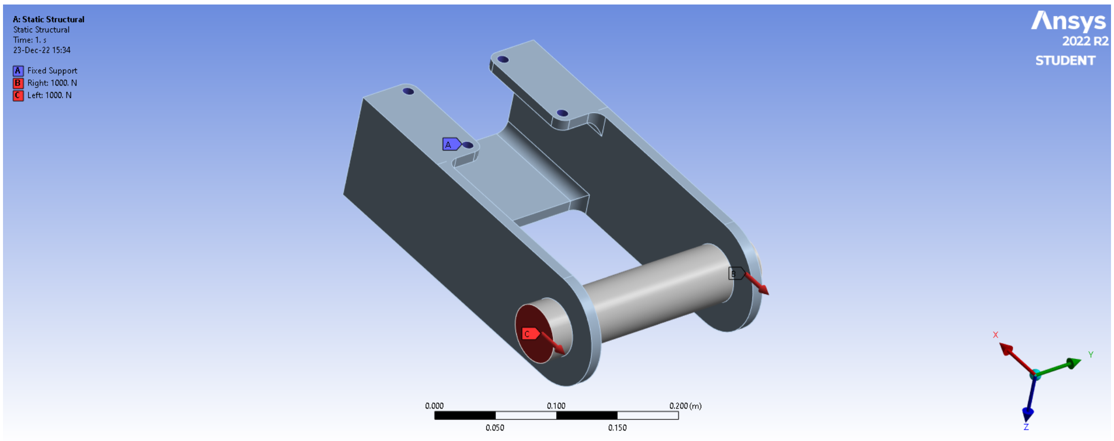


Figure 5. The boundary conditions and the applied forces within the system

The second step was to solve the simulation for 8 different cases and get the results. Before introducing the other cases to the simulation, we first solved the first case with the given arrangements and seek for the von Misses/Equivalent Stress along the bracket body. The maximum stress value along the body is again parametrized so that the parametrization is complete with 6 inputs of force and direction and output of maximum equivalent stress.

The final step was to post-process and interpret the results such that the difference between the loading scenario and the safety factor is imminent. The first loading scenario result can be seen in Figure 6.

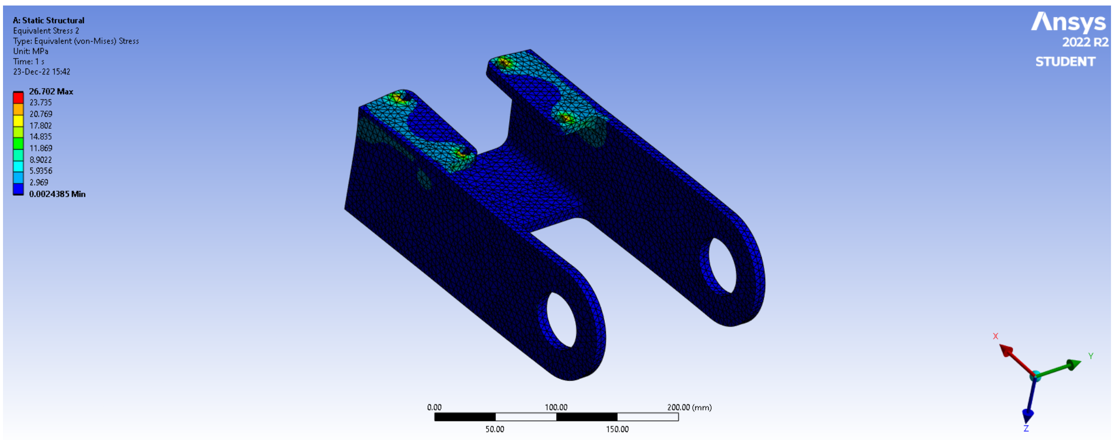


Figure 6. The equivalent stress contour plot of the first loading scenario

The detailed analysis of the results will be explained in the results section, the safety factors are computed and the findings are demonstrated.

Results

With the parametrization tool, the 8 different loading scenarios were introduced to ANSYS, where the inputs are the forces in certain directions with different magnitudes. Note that the total loading magnitude on the pin faces sums up to 1 kN but the angle changes therefore the component magnitudes differ in magnitude and sign. After solving for the cases one by one, the maximum equivalent stress values are logged into Table 1, alongside the input values.

Table 1. The input and output values of the problem

	A	B	C	D	E	F	G	H
1	Name	P1 - Right X Component	P2 - Right Y Component	P3 - Right Z Component	P4 - Left Y Component	P5 - Left Z Component	P6 - Left X Component	P7 - Equivalent Stress 2 Maximum
2	Units	N	N	N	N	N	N	MPa
3	DP 0 (Current)	-1000	0	0	0	0	-1000	26.702
4	DP 1	1000	0	0	0	0	-1000	18.213
5	DP 2	0	0	-1000	0	-1000	0	138.02
6	DP 3	0	0	1000	0	1000	0	138.02
7	DP 4	0	0	1000	0	-1000	0	99.853
8	DP 5	-707.1	0	-707.1	0	-707.1	-707.1	113.76
9	DP 6	707.1	0	707.1	0	-707.1	-707.1	81.801
10	DP 7	-707.1	0	707.1	0	-707.1	707.1	65.397

The safety factor can be calculated using the formula given in the textbook [1],

$$SF = \frac{\sigma_{yield}}{\sigma_{von\ Misses}}$$

Table 2. The calculated safety factors of each loading case where $\sigma_{steel} = 250\ MPa$.

	Maximum Equivalent Stress (MPa)	Safety Factor
Case 1	26.70	9.36
Case 2	18.21	13.73
Case 3	138.02	1.81
Case 4	138.02	1.81
Case 5	99.85	2.50
Case 6	113.76	2.20
Case 7	81.80	3.06
Case 8	65.40	3.82

Discussion/Comparison with Literature

The pin-bracket system was loaded with the same magnitude in all 8 cases, although the loading scenario was different for each case. The difference in the loading changes the stress accumulation within the bracket due to changing normal stress, shear stress, and bending moment. When the results have been investigated the region of maximum stress is always on the boltholes of the bracket due to the stress concentration. As can be seen in Figure 7, the maximum stress occurs on the side of the bolt holes. Due to the limitations of our mesh, the max stress seems to be on the mesh nodes of the tetragonal elements but in actuality, they don’t specifically lay on those points.

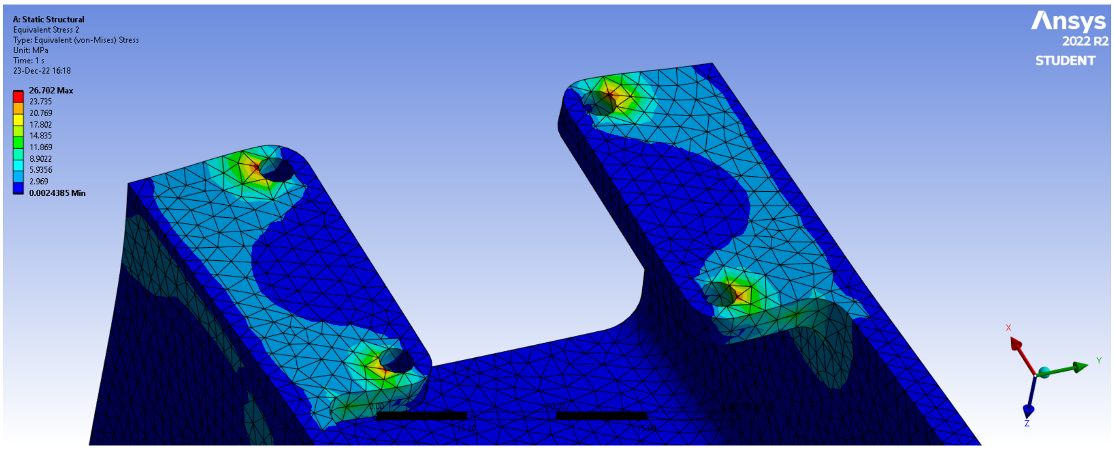


Figure 7. The region of maximum equivalent stress on the bracket

When the loading scenarios are compared the maximum safety factor is achieved when the in the 2nd case, where the loadings are applied in opposite directions and purely in the x-axis, this balances out both the total moment and forces in most of the regions within the body, therefore, yielding a very safe system. The minimum safety factor, on the other hand, was achieved in the 3rd and 4th cases, where the loadings are done in the same direction on both sides and on the z-axis. This maximizes the bending moment in the vicinity of the boltholes, therefore yielding the maximum local stresses.

The results are compelling in the sense that the maximum stresses occur at the boltholes due to the stress concentration, and general engineering knowledge [1]. Since the material used in the study is ductile these sorts of local stresses wouldn’t necessarily pose a problem since local yielding would marginalize the high stresses, yet stress concentrations as low as 1.81 could be a problem depending on the application.

List of References*

[1] R. C. Juvinall and K. M. Marshek, Fundamentals of Machine Component Design Global Edition, SI version. Wiley, USA, 2020.