

RESEARCH ARTICLE / ARAŞTIRMA MAKALESİ

STATIC STRENGTH AND HARMONIC VIBRATION ANALYSIS OF THE SPIDER CRANE BY
COMPUTATIONAL MECHANICS SOFTWARETacetdin SEVDİM¹¹Altınbaş University, School of Engineering and Natural Sciences, Mechanical Engineering, Istanbul.
tacetdin.sevdim@gmail.com ORCID No: 0000-0002-8094-6167Orhan Koray ÇALIK²²Altınbaş University, School of Engineering and Natural Sciences, Mechanical Engineering, Istanbul.
koray_calik@windowslive.com ORCID No: 0000-0001-6704-4970Süleyman BAŞTÜRK³³Altınbaş University, School of Engineering and Natural Sciences, Mechanical Engineering, Istanbul.
suleyman.basturk@altinbas.edu.tr ORCID No: 0000-0002-9016-8397

Received Date/Geliş Tarihi: 30/01/2019. Accepted Date/Kabul Tarihi: 17/05/2019.

Abstract

In this study, it is aimed to investigate and perform the static structural and natural frequency analysis of the mini spider crane which is designed to lift 0-500 kg load and 0-3 meters extend range, by using computational mechanics software. The crane works and driven by hydraulic system and in order to get the more realistic result, hydraulic cylinders modeled as springs instead of stiff rigid bodies in the analysis. This simulation approach provides a different method than the conventional method. Detailed analysis show that the spider crane statically can lift 1200 kg before the plastic region, so its safety factor is 2.4 when it is lifting 500 kg and its natural frequencies are 16.2 Hz and 50 Hz.

Keywords: Spider Crane, Static Analysis, Hydraulics, Finite Elements, Computational Mechanics, Design.

**HESAPLAMALI MEKANİK YAZILIMIYLA ÖRÜMCEK VİNCİN STATİK MUKAVEMET VE
HARMONİK TİTREŞİM ANALİZLERİNİN YAPILMASI****Öz**

Bu çalışmada, 0-3 metreden 0-500 kg yük kaldırmak üzere tasarlanmış bir mini örümcek vincin statik yapısal ve doğal frekans analizlerinin araştırılması ve hesaplamalı mekanik yazılımı kullanılarak uygulanması hedeflenmiştir. Hidrolik sistemle çalışan ve tahrik edilen vincin hidrolik silindireleri daha gerçekçi bir sonuç almak için katı bir cisim yerine yay olarak modellenmiştir. Bu simülasyon yaklaşımı geleneksel metoda göre farklı bir metot sağlamaktadır. Detaylı analizler sonucunda, örümcek vincin plastik bölgeye geçmeden en fazla 1200 kg yük kaldırabileceği yani 500 kg yükte iken güvenlik katsayısının 2.4 olduğu, doğal frekansının ise 16.2 Hz ve 50Hz olduğu saptanmıştır.

Anahtar Kelimeler: Örümcek Vinç, Statik Analiz, Hidrolik, Sonlu Elemanlar, Hesaplamalı Mekanik, Tasarım.

1. INTRODUCTION

Engineering software programs are developing every day and they provide to design, analysis and manufacture complex machines and structures. Also, thanks to Finite Element Methods, complex mathematical problems can be solved in a short time by the computers and it provides us to model, simulate and analyze the mechanical system on the computer by using minimum material and sources.

In this study, a mini spider crane was modeled and analyzed under the different loading conditions and maximum load capacity has been obtained with a realistic approach by using ANSYS software. Furthermore, natural frequency of the crane was obtained to avoid resonance failures by using modal and harmonic response analysis. In order to get the realistic results, hydraulic cylinders modeled as springs because as Drexler indicates that hydraulic oil is compressing under the high pressure and acting like a spring instead of transmitting the force directly like a rigid body and also damping the vibration (Drexler et al., 1988; Sochacki and Bold, 2016; Bold et al., 2018).

Computational engineering software programs also provide us to realize the weak and strong parts of the design before the manufacturing phase so that we can re-design and make the necessary changes to obtain an optimum model as made in this study as well.

Another advantage of the software is that it allows us to change the input variables like loading forces and it calculates the outputs such as total stress and deformation values parametrically, so solution envelope of the model could be obtained easily.

2. RESEARCH OBJECT AND MATERIAL

The Spider Crane which used for analysis was designed for “Design and Manufacture of Mini Spider Crane” project by using Autodesk Fusion 360 CAD program. The Crane is designed to carry 500 kg load when the maximum extend is 3 meters and It has 3 booms which one of them is knuckle and another one is telescopic. The booms are made up of Domex700, high strength low alloy steel, with 700 MPa yield strength and 210 GPa Young’s modulus (Ahmad, 2011).

The main parts of the spider crane can be seen in Figure 1 as booms, chassis, slewing drive, outriggers, solid tires and hydraulic cylinders. Also, the booms and the cylinders can be seen with their numbers in Figure 2 and general dimensions of the spider crane can be found in Figure 3.

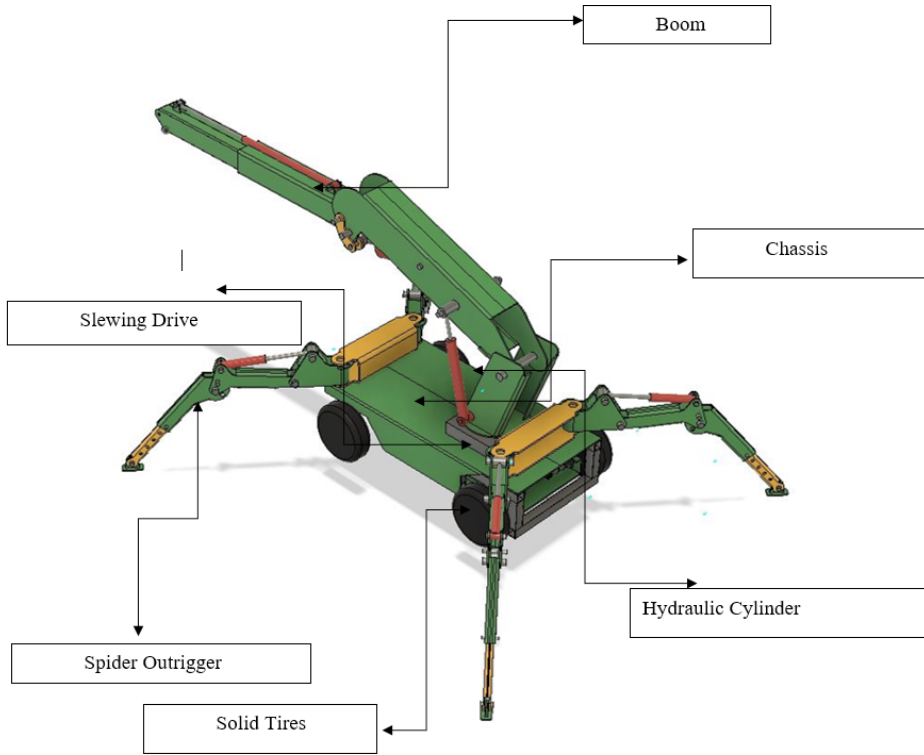


Figure 1. Isometric view of the spider crane.

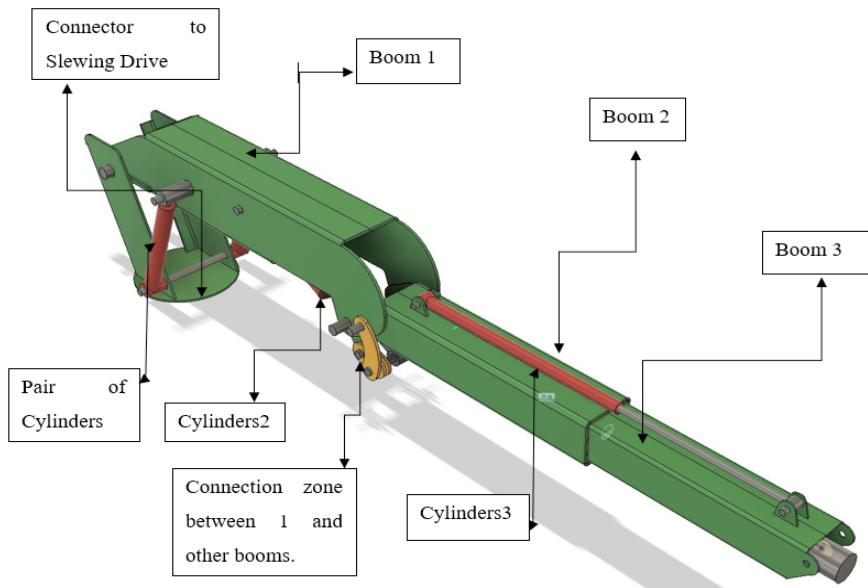


Figure 2. Booms of the Spider Crane.

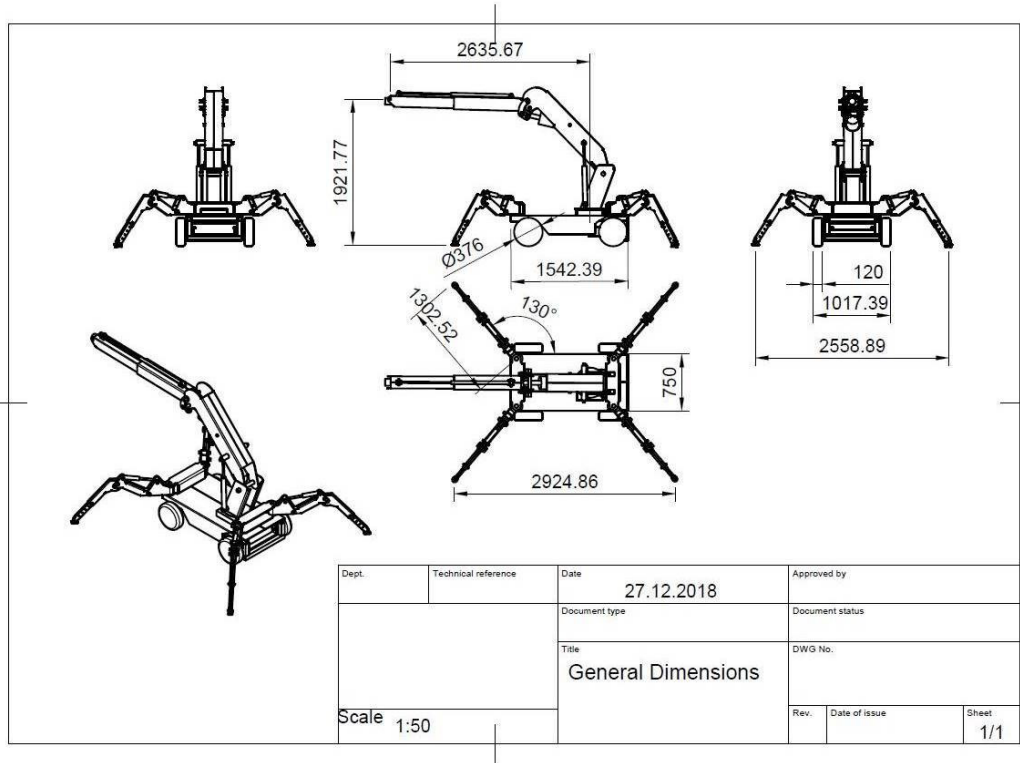


Figure 3. General dimensions of the spider crane.

3. STATIC STRUCTURAL ANALYSIS

The steps of the static structural analysis can be divided into these parts such as selecting the material, choosing the connection and/or joint types, if more than one body exist in the analysis, meshing the bodies and finally setting the conditions of the analysis which is obtaining that from where the structure will be fixed and from where the force is applied.

3.1 Static Analysis For Booms:

3.1.1 Choosing the Joint and Connection Types Between the Bodies of the Model

Choosing accurate joint/connection type is very crucial step for the static structural analysis. For example, when the connection type between the booms are “bonded” (acts like a rigid body) instead of the “revolute” joint, very high stress results occurred around the joint elements (ANSYS Inc., 2013). In order to obtain the more realistic results, joints and connections were chosen as seen in Figures 4-7. Figure 4 shows the frictionless connection between the first boom and body and revolute joint between the first boom and the shaft because the first boom is turning around the shaft. Figure 5 and 6 show the revolute joints around the shafts as well. Figure 7 shows the telescopic connection as translational joint between the second and third booms.

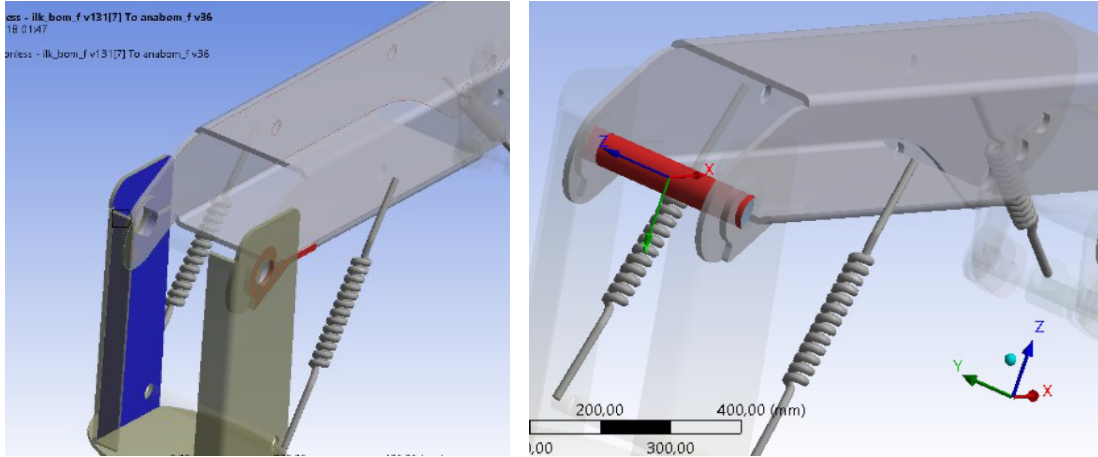


Figure 4. Frictionless connection between walls of the first boom and body (Left), Revolute joint between the first boom and shaft at the body (right).

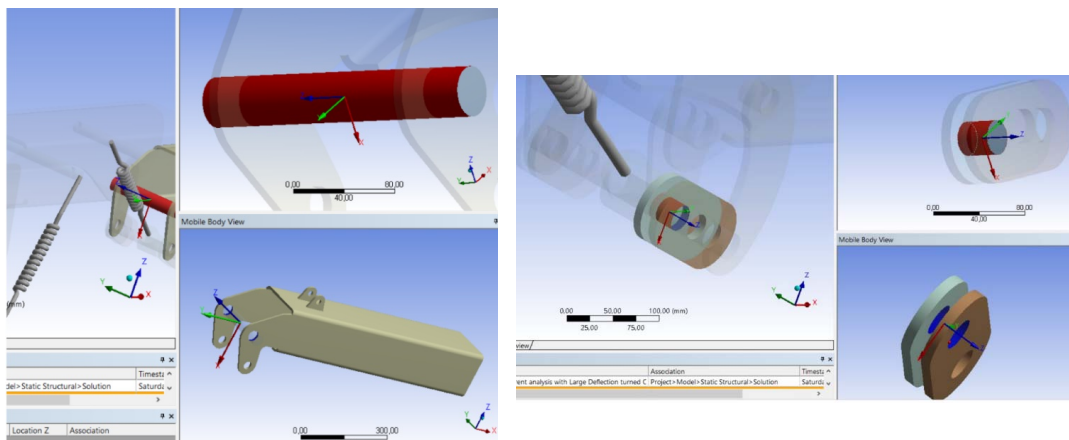


Figure 5. Revolute joint between the second boom and shaft at the first boom (left), Revolute joint between the support parts which connects the first and the second booms and the connection shaft between them (right).

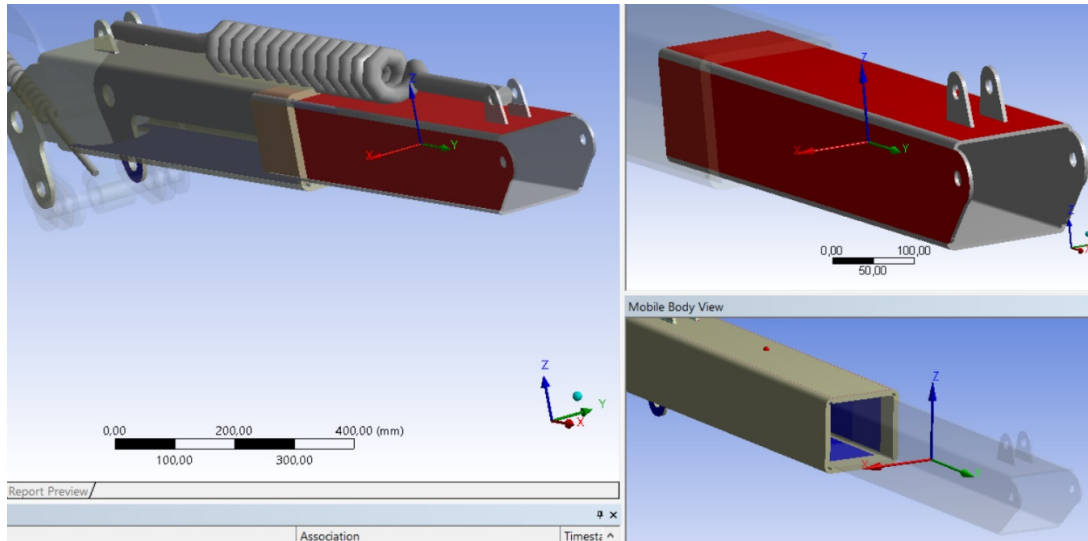


Figure 6. Translational joint between the second and third booms.

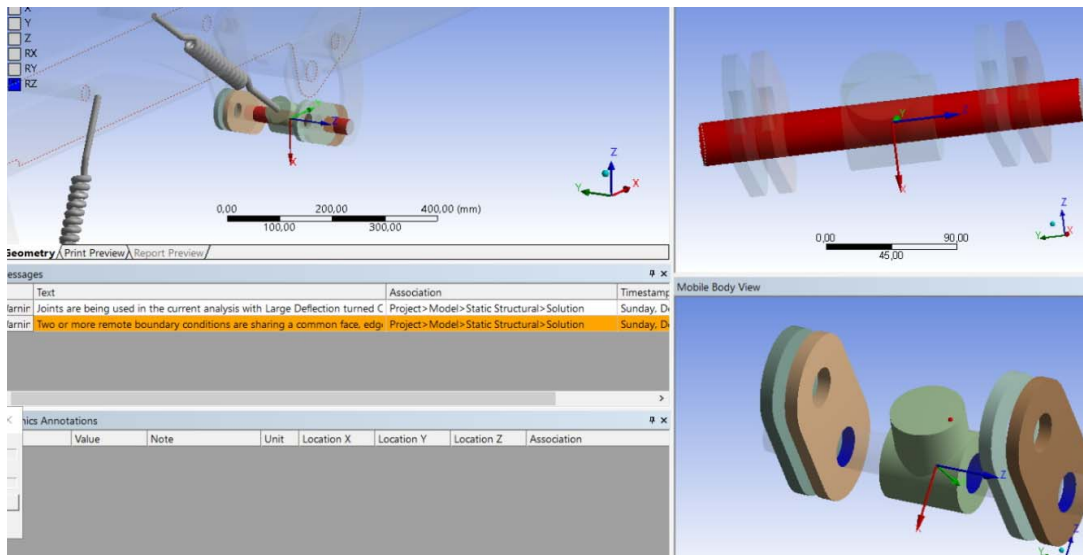


Figure 7. Revolute joint between the support parts, second cylinder connection part and shaft between second and third booms.

3.1.2 Setting the Conditions of the Static Structural Analysis

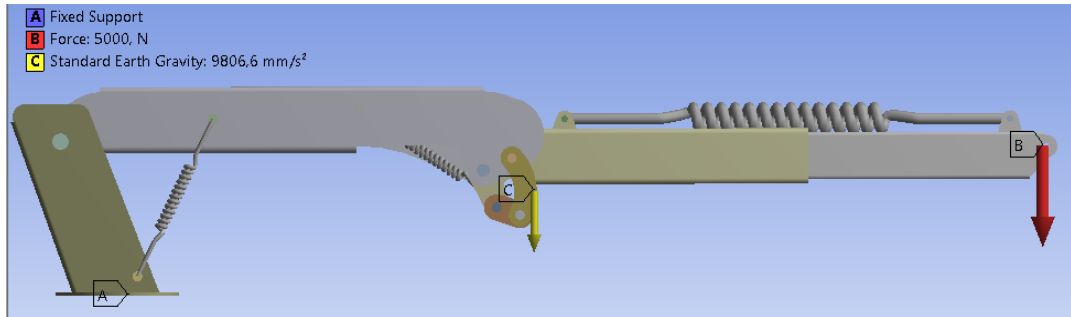


Figure 8. Static structural of the model.

First of all, the model has to be fixed in the space that is why booms are fix supported at point A which is a connection surface between the body and the chassis as seen in Figure 8. Secondly, downward load applied from the holes at the end of the third boom which can be seen in Figure 8 at point B as 5000 N load. Also, Standard Earth Gravity accounted into the analysis to calculate booms' own weight as well.

3.1.3 Von Misses Stress Distribution Results

The Maximum Stress will be occurred when the crane is working at zero degree, where it is the farthest point and therefore, the moment will be the maximum. Results of the analysis seen in Figure 9 show that while 5000 N load applied at the end of the third boom, the maximum stress occurred around the shaft between the first boom and the body as 328 MPa, which means approximately half of the 700 MPa Yield strength and the factor of safety is equal to 2.1. Also note that, the crane is tested with 500 kg load but for continuous work it will be better to use with 250-300 kg load so, safety factor will be 3-3.5 for continuous working conditions.

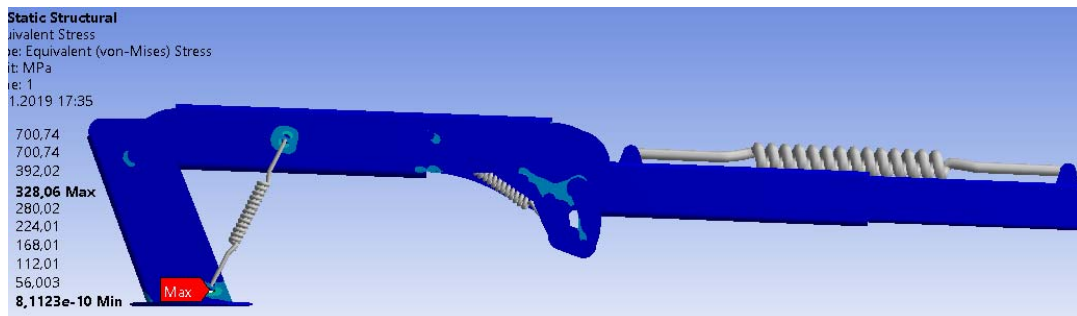


Figure 9. Von misses stress distribution because of 500kg load.

3.1.4 Total Deformation Results

We could obtain the amount of the elongation from the deformation results. Three important point can be drawn from Figure 10, first cylinder is compressed almost as calculated as in the “modelling a hydraulic cylinder as a spring”. There is around 60 mm difference between the connections of the third cylinder. It shows relatively high deformation on the third cylinder, in order to avoid from bending situation, some hydraulic oil can be kept in front of the piston.

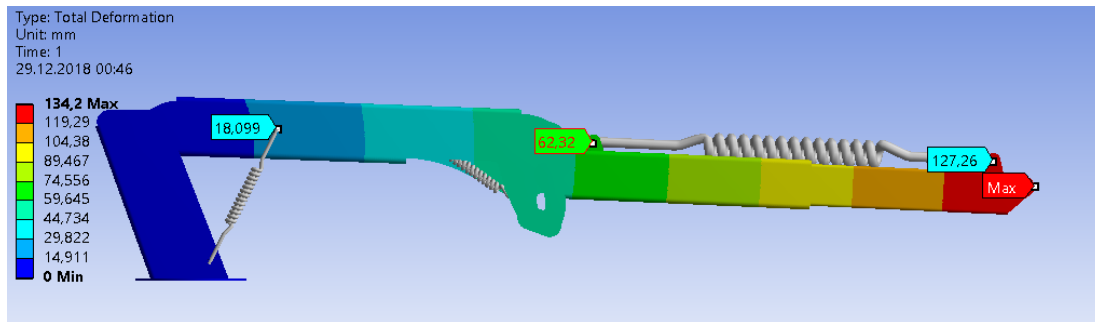


Figure 10. Total deformation because of 500 kg load.

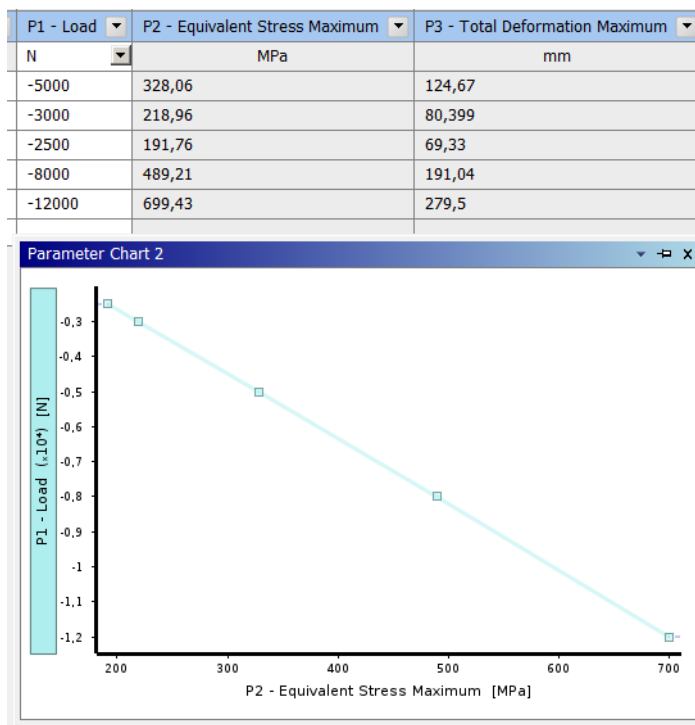


Table 1. shows the values and graph of Load versus- Stress/Deformation chart for different loading conditions.

Parametric analysis allows us to see the difference between Stress/Deformation results under the different loading conditions. This parametric result shows that in the worst situation for load of booms' position (minimum angle and furthest elongation working), Crane can carry max 1200 kg statically while factor of safety is equal to 1 and just before the passing through the plastic region.

3.2 Static Analysis For Outriggers:

The maximum load on the foots will be occurred in the position as seen in Figure 11. From moment calculation below, max load for one foot for 500 kg was found as 11 kN.

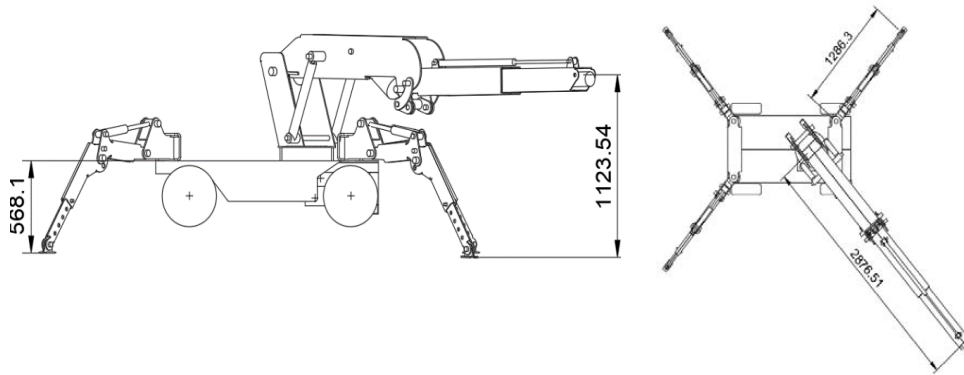


Figure 11. The maximum load on the foots.

3.2.1 Setting the Static Structural of the Outrigger Model

Foot is fix supported at point B which has the connection surface to the chassis and 11 kN force is applied at point A as seen in Figure 12.

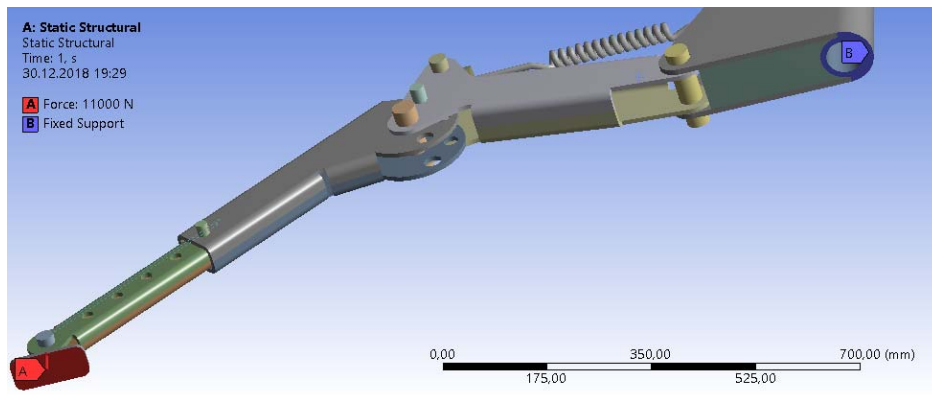


Figure 12. Static structural of the model.

3.2.2 Von Misses Stress Distribution Results

Figure 13 and 14 show that the maximum stress occurred only around the one point which is located next to the fix supported area as shown as point B in Figure 14.

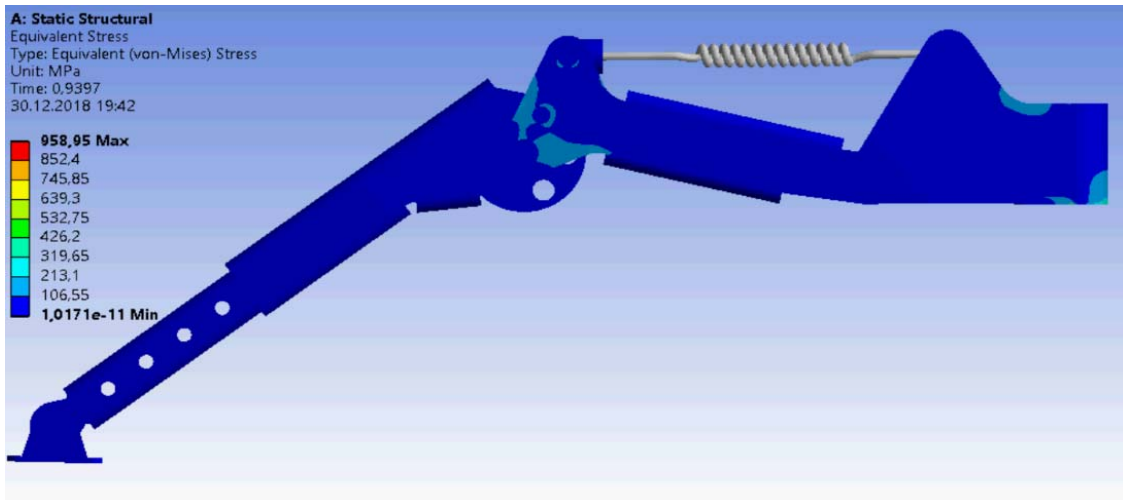


Figure 13. Von misses stress distribution because of 500 kg load, 11 kN.

When it is zoomed to the vertex which maximum stress occurred and checked the stress distribution next to this point shows us that the actual stress is relatively high because of supporting method. That is why we can ignore this point and assume that we are in safe region but in the limits.

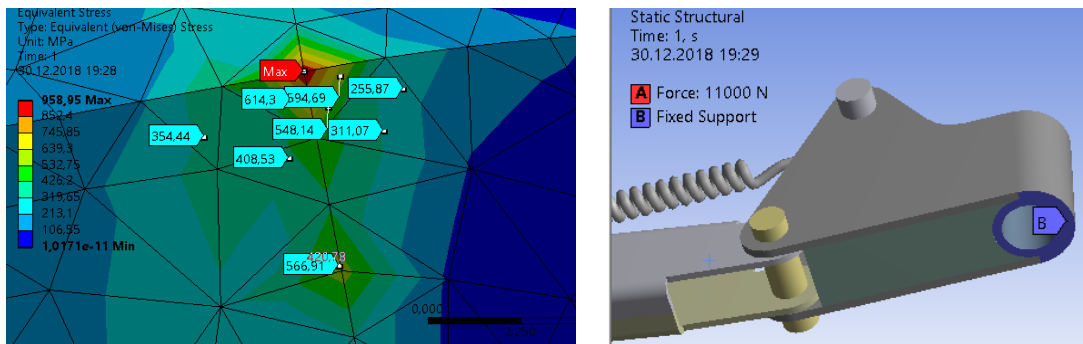


Figure 14. Maximum stress area (Left), Maximum stress occurred because of fix supported face (right).

3.3 Modelling a Hydraulic Cylinder as a Spring

Hydraulic cylinders are acting like more a spring than a rigid body under the high pressure (especially 150 Bar and above). Drexler et al., 1988 says that “in the case of a fluid containing no air bubbles the volume

reduces by 0.7% when the pressure is increased by 100 bar". According to this ratio, volume will reduce by 0.14% with 200 Bars.

Calculating Spring Coefficients:

$$\text{Area of the piston is defined as } A = \frac{\pi \cdot d^2}{4} \quad (1)$$

$$\text{Volume of the cylinder} = \text{Area} \times \text{Stroke} = V = A * S \quad (2)$$

$$\text{Compressed volume under 200 Bar is } V_c = V * 0.0014 \quad (3)$$

$$\text{Compressed Stroke is Compressed Volume / Area} = S_c = \frac{V_c}{A} \quad (4)$$

$$\text{Stiffness coefficient of the spring is } k = \frac{F}{x} = \frac{F}{S_c} \quad (5)$$

When the first cylinder is completely open, volume of the first cylinder is 0.46 L, stroke is 370 mm and Diameter is 40 mm, so the stiffness coefficient of the spring is calculated as $k=1290 \text{ N/mm}$.

Ansys used for testing the stiffness coefficient (k) of the spring which is obtained from the above mentioned calculation. First cylinder's piston diameter and stroke length modeled with the spring and the same pressure applied to the first piston (200 Bar = 20 MPa). Also, gravitational acceleration was applied.

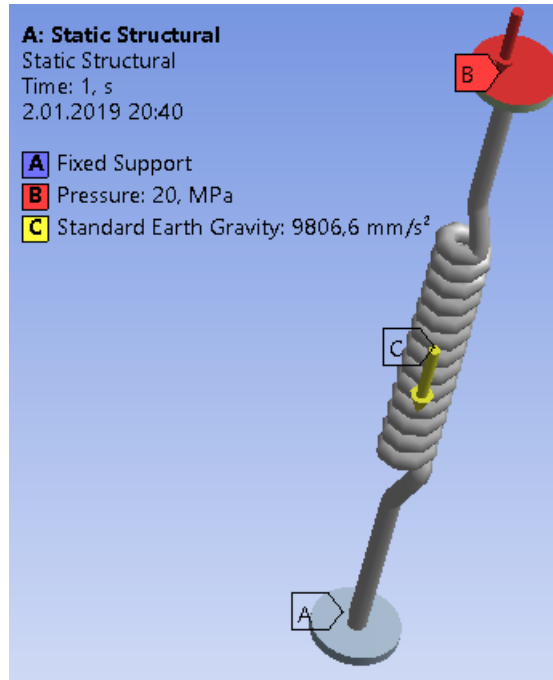


Figure 15. Static structural set.

Material	None
Type	Longitudinal
Spring Behavior	Both
Longitudinal Stiffness	1290, N/mm
Longitudinal Damping	0, N-s/mm
Preload	None
Suppressed	No
Spring Length	370, mm

Table 2. Properties of the spring.

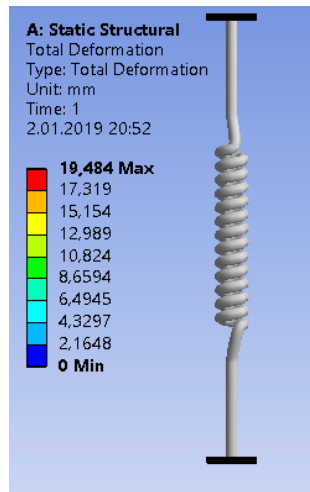


Figure 16. Directional deformation of the spring.

This test shows us that springs can be used as hydraulic cylinders. Instead of having more realistic result another advantage of using springs that we do not have to mesh bodies of the cylinders, so solving time and the number of elements are reduced (Rodriguez et al., 2017). On the other hand, when we do not use the cylinders as a body, their weights are not accounted in the analysis. Note that when the first cylinder is closed (0 degree and maximum elongation horizontally) stroke is 4.7 cm. Spring coefficient of cylinders are calculated by using the values shown in Table 3.

Cylinder	Piston Area (cm ²)	Stroke (cm)	Volume (cm ³)	Compressed Volume (cm ³)	Compressed Stroke(cm)	Force (N)	Spring Coefficient (N/mm)
1.	12,56	4,7	59	0,0826	0,06583	21040	319594
2.	78,54	30	2356,19	3,29	0,42	144050	342976
3.	39,4	57	2245	3	0,79	7350	9303

Table 3. Values which are used to calculate the spring coefficients for each cylinder.

4.HARMONIC RESPONSE ANALYSIS

Harmonic Response Analysis shows us that how and in which directions the structure is responding when the natural frequency values are applied to the overall assembly. It is known that vibration plays a crucial role for fatigue and crack failures. In order to get Harmonic Response results firstly natural frequency intervals were found from Modal Analysis and then by using these interval, natural frequency can be found from the harmonic response analysis. Sochacki and Bold explain that the hydraulic cylinders are damping the vibration by an important amount (Jiang et al., 2016; Feng et al., 2017). That is why, using the springs for simulate the hydraulic cylinders provided a more realistic result for harmonic response analysis as well.

	Mode	Frequency [Hz]
1	1,	8,7561
2	2,	16,415
3	3,	35,069
4	4,	39,084
5	5,	50,925
6	6,	92,46
7	7,	100,1
8	8,	108,73
9	9,	125,83

Table 4. Natural frequency interval found from Modal Analysis.

As seen from Table 2, we are interested in first 5 modes which generally electrical motors work 0-3000 rpm equals to 0-50 Hz. That is why in harmonic response analysis 0-60 Hz interval accounted.

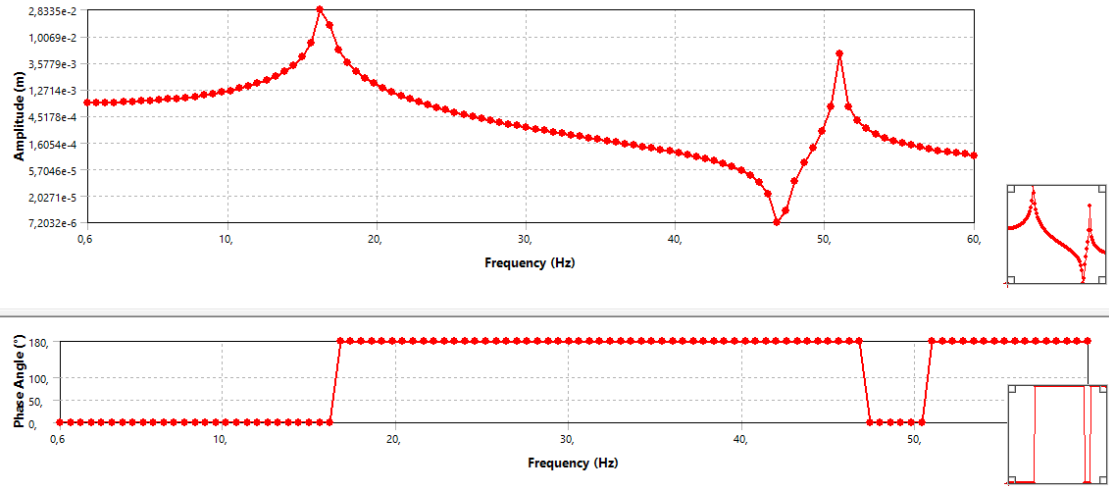


Figure 17. Natural frequency interval founded from harmonic response analysis.

	Frequency [Hz]	Amplitude [Pa]	Phase Angle [°]		Frequency [Hz]	Amplitude [Pa]	Phase Angle [°]
1	0.6	49337	180.	50	30.	12979	0.
2	1.2	49520	180.	51	30.6	11799	0.
3	1.8	49829	180.	52	31.2	10702	0.
4	2.4	50269	180.	53	31.8	9675.	0.
5	3.	50846	180.	54	32.4	8709,6	0.
6	3.6	51572	180.	55	33.	7797,1	0.
7	4.2	52459	180.	56	33.6	6930,5	0.
8	4.8	53524	180.	57	34.2	6106,4	0.
9	5.4	54788	180.	58	34.8	5354,1	0.
10	6.	56278	180.	59	35,4	4446,2	0.
11	6.6	58029	180.	60	36.	3726,9	0.
12	7.2	60084	180.	61	36.6	2994.	0.
13	7.8	62497	180.	62	37,2	2270,4	0.
14	8.4	65319	180.	63	37,8	1563,6	0.
15	9.	68871	180.	64	38,4	916,51	0.
16	9.6	72882	180.	65	39.	1740,6	0.
17	10.2	77810	180.	66	39,6	1153,7	180.
18	10.8	83881	180.	67	40,2	1809,7	180.
19	11.4	91490	180.	68	40,8	2617.	180.
20	12.	1,0126e+005	180.	69	41,4	3501,2	180.
21	12.6	1,1419e+005	180.	70	42.	4462,2	180.
22	13.2	1,3205e+005	180.	71	42,6	5514,1	180.
23	13.8	1,5823e+005	180.	72	43,2	6679,3	180.
24	14.4	2,0015e+005	180.	73	43,8	7989,2	180.
25	15.	2,7779e+005	180.	74	44,4	9486,8	180.
26	15.6	4,7003e+005	180.	75	45.	11233	180.
27	16.2	1,7356e+006	180.	76	45,6	13316	180.
28	16.8	9,4549e+005	0.	77	46,2	15867	180.
29	17.4	3,5999e+005	0.	78	46,8	19097	180.
30	18.	2,1792e+005	0.	79	47,4	23355	180.
31	18.6	1,5395e+005	0.	80	48.	29281	180.
32	19.2	1,176e+005	0.	81	48,6	38172	180.
33	19.8	94178	0.	82	49,2	53131	180.
34	20.4	77836	0.	83	49,8	83880	180.
35	21.	65794	0.	84	50,4	1,8461e+005	180.
36	21.6	56555	0.	85	51.	1,3172e+006	0.
37	22.2	49246	0.	86	51,6	1,5017e+005	0.
38	22.8	43320	0.	87	52,2	81121	0.
39	23.4	38418	0.	88	52,8	56181	0.
40	24.	34296	0.	89	53,4	43278	0.
41	24.6	30781	0.	90	54.	35369	0.
42	25.2	27746	0.	91	54,6	30010	0.
43	25.8	25098	0.	92	55,2	26128	0.
44	26.4	22765	0.	93	55,8	23180	0.
45	27.	20693	0.	94	56,4	20860	0.
46	27.6	18837	0.	95	57.	18982	0.
47	28.2	17164	0.	96	57,6	17428	0.
48	28.8	15644	0.	97	58,2	16119	0.
49	29.4	14256	0.	98	58,8	14998	0.
50	30.	12979	0.	99	59,4	14027	0.

Table 5. Natural frequency interval found from harmonic response analysis

Table 5 and Figure 17 show that 16.2 Hz (around 1000 rpm) and 50 Hz (3000 rpm) are natural frequencies for this structure. That is why, electrical motor should not work continuously exactly in these rpms. 1500 rpm (25 Hz) can be chosen as a safety area.

5. CONCLUSION

In this study, loading capacity and natural frequencies of the spider crane are obtained and results of the analysis enabled us to claim where the maximum stress occurred and in which frequencies the structure have the natural frequencies. Selecting the appropriate connection/joint types between the bodies and modelling the hydraulic cylinders as the springs are crucial steps for this kind of analysis which has many connections and joint parts. According to the static structural analysis results, strokes of the cylinders are optimized in the design process to avoid bending failures, because of high deformation values especially on the third boom. Furthermore, harmonic response analysis' results enabled us to determine the working rpm of the electrical motor in order to avoid from fatigue or crack failures based on the resonance problems. The natural frequency of the model can be changed by adding some supports or changing the material/thickness in the design if it is necessary.

This study can be a reference for the future studies such as reducing the vibration at the end of the booms' of the cranes or optimizing the model of the booms and the cranes.

Conflict of Interests/Çıkar Çatışması

Authors declare no conflict of interests/Yazarlar çıkar çatışması olmadığını belirtmişlerdir.

6. REFERENCES

- H. Feng, Q. Du, Y. Huang and Y. Chi.** 2017. "Modelling Study on Stiffness Characteristics of Hydraulic Cylinder under Multi-Factors", *Journal of Mechanical Engineering*.
- M. Bold, S. Garus and W. Sochacki.** 2018. "Damped Vibrations Of Telescopic Crane Boom", *24th International Conference Engineering Mechanics 2018, Svratka*.
- Nd.** 2013. "ANSYS Mechanical APDL Contact Technology Guide", ANSYS Inc., Southpointe.
- P. Drexler, H. Faatz, F. Freicht and F. Freicht.** 1988. "Planning and Design of Hydraulic Power Systems", Main: Mannesmann Rexroth AG.
- S. M. Ahmad.** 2011. "Study of Dynamic Behaviour of Multi-Layered Structures Subjected to Blast Loading".
- W. Jiang, X. Luo and X. Chen.** 2016. "Influence of structural flexibility on the nonlinear stiffness of hydraulic system", *Advances in Mechanical Engineering*. Vol 8, pp 1-7.
- W. Sochacki and M. Bold.** 2016. "Transverse and Longitudinal Damped Vibrations of Hydraulic Cylinder in a Mining Prop", Vol 27

V. G. Rodríguez, J. J. C. Eras, N. V. Izquierdo, H. H. Herrera and A. S. Gutierrez. 2017. "Calculation Model And Methodology For Stiffness Evaluation In Hydraulics Cylinders", *ARPN Journal of Engineering and Applied Sciences*. Vol 12, 1361.