

CONTACT PATTERN INVESTIGATION OF GEAR DRIVES USING FINITE ELEMENT METHOD

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Abstract: This article deals with a study that examines deviations in the curve generated by CAD software and real involutes. As part of this, FEM tests are carried out further from the conclusions drawn from the deviations of the gears created, the purpose is to determine the contact pattern at the design stage. Furthermore, using the results of the studies, we can determine how accurately our results give an accurate picture compared to the result that can be calculated from the theory. Finally, we also test for shafts deformation in the connection of the gears.

Keywords: *gears, CAD model, FEM analysis, mounting defects*

1. INTRODUCTION

Gear drives cover a very significant area in traditional mechanical engineering. The aim of the authors of this article is to create a method based on CAD and FEM software, with which the examination of the contact pattern of gear drives can be carried out during the design phase. For each FEM test, it is necessary to create a CAD model. In this article, we will briefly summarize the shape difference of the toothed elements generated by CAD software compared to the theoretical (involute) tooth profile. From this we make a decision whether these gears can be used for FEM testing. If the difference is too large, we only get a right result during the FEM test if we create the gear ourselves. After that, we create a CAD model that presents a gear drive. Steps to create this proper gear are described in detail. In addition, we perform the FEM test and determine how much the results obtained give a true picture compared to the theoretical values described in standards. We also create a model that shows a gear drive with shafts, and we also carry out tests on this to see what problems the deformation of shafts can cause during real operation.

2. COMPARISON OF TOOTH PROFILES OBTAINED FROM CAD SYSTEM WITH THEORETICAL INVOLUTE [1]

CAD systems currently in use have a predefined program module for making built-in toothed elements. However, we do not know how realistic the gears generated in this way are and how much the curve describing the tooth profile corresponds to the theoretical involute. Thus, we need to consider whether they are only for visual display or give back the tooth profile completely correctly. It is determined that for the test we will use a gear with a 20 tooth number and a module of 3 mm [1]. The CAD system we use is Solid Edge, in this software the gear model is made. We also used an MS excel table to draw, in which we calculate each control points of the involute curve. The finished model is shown in *Figure 1*. Here you will also see the construction lines used for drawing, which are made up of the geometry of the gear.

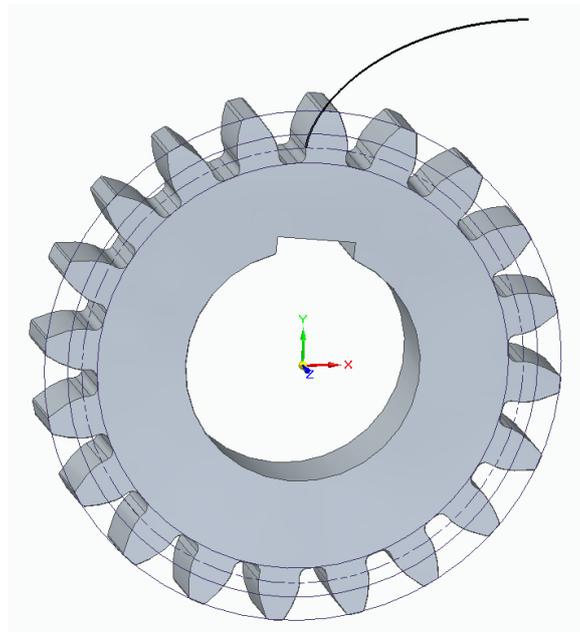


Figure 1. The model of the finished gear [1]

After that, with the help of the built-in module, we create the generated gears. For this we also use Solid Edge. To have a different software, we also create the gear with Inventor's module. We make tests on the tooth profile, with visual inspection in first approach. Here we place the teeth side by side in a comparison picture, covering them with the gear we have drawn. The result is illustrated in *Figure 2*, which shows the differences in the different tooth profiles of different softwares. In the figures, the grayed tooth profile shows the tooth of the gear we generate, and the green color shows the profile generated by the software.

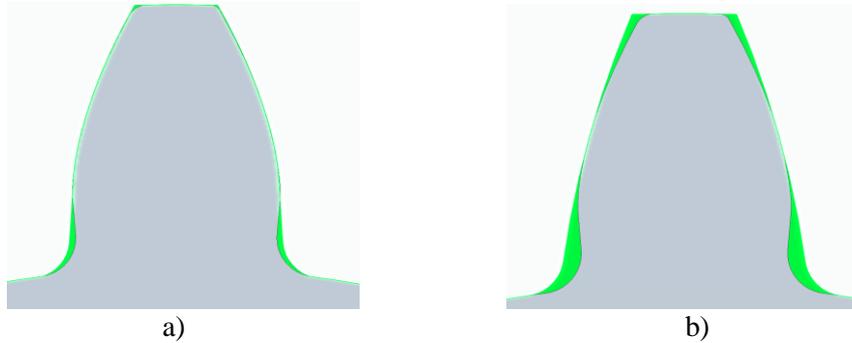


Figure 2. a) Comparison-Solid Egde, b) Comparison-Inventor

As shown in *Figure 2*, there are differences between the tooth profiles of the gears. The values of this can also be determined [1]. Furthermore, we can perform the test by changing the values of the module and examining the differences in these gears. In our case, we create gears ranging from a 1 mm module to a 5 mm module, on which we measure the deviation values at the topline and the pitch circle. The measured deviations for gears created in Solid Edge are shown in *Table 1*. We only noticed a deviation at topline, so we can see its percentage evaluation.

Table 1
Differences in gears generated by Solid Edge

Module value	Drawn by us	CAD generated	Variance
1 mm	0.693 mm	0.701 mm	1.15%
2 mm	1.386 mm	1.403 mm	1.22%
3 mm	2.075 mm	2.105 mm	1.44%
4 mm	2.773 mm	2.806 mm	1.19%
5 mm	3.465 mm	3.508 mm	1.24%

The difference between the gear created by Inventor and the wheel we create is summarized in *Table 2*. It must be said here that, as in the previous case, there was only a difference in the point taken at the topline.

Table 2
Differences in gears generated by Inventor

Module value	Drawn by us	CAD generated	Variance
1 mm	0.693 mm	0.869 mm	25.3%
2 mm	1.386 mm	1.727 mm	24.6%
3 mm	2.075 mm	2.591 mm	24.8%
4 mm	2.773 mm	3.454 mm	24.5%
5 mm	3.465 mm	4.318 mm	24.6%

Here the question arises as to what is the difference that can still be allowed if we want to carry out a finite element analysis with them. Both models may be available for some tasks, but a small deviation in Solid Edge may be a problem in other tests. We have to examine this further to how much deviation may affect our results. In conclusion, it can be said that if we already know the process of the solution, how to draw involute curves from calculated results in CAD systems, and we have such experience, it may be better to design and produce the gear model by us, because this way we will surely get the right result.

3. PREPARATION OF GEARS IN CAD SYSTEM FOR FEM TESTING

In order to use the CAD model to test contact surfaces, we need to rotate the two gears. Here we use the gears we make. The next step is that to position the two gears, you will need an auxiliary part. In their case, it will be a plate, which will also be drawn in Solid Edge. In the ISO metric assembly module, we load in our drawn plate. On the plate we apply the center distance of the two gears, based on the calculation 60 mm. At a distance corresponding to the center distance, a hole is drawn on the plate. If this is the case, we will insert two gears into our assembly. Use the *Add Component* icon here. When we were done with this, we will start to put the parts together with the help of constraints. On the plate, using the *Mate* command and selecting one of the planes of the gear, we fit it to the plane of the plate. This puts them on the same plane, but in order to be in the correct hole, use the *Insert* command. Selecting the center line of the hole and the center line of the gears creates a connection between them. The finished model is shown in *Figure 3*. A parallel keyway has also been drawn in the holes of the gears to make the gear position more noticeable during rotation.

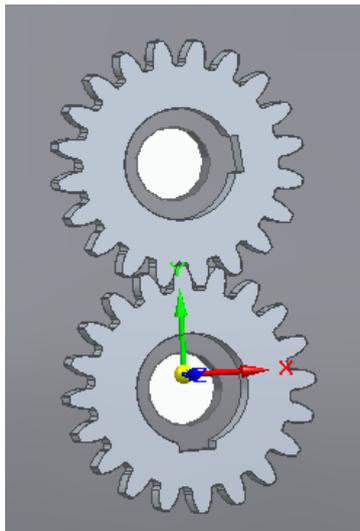


Figure 3. Finished assembly in Solid Edge

In the finished assembly, we still need to find the tooth connection in order to simulate physical operation and thus save more CAD models for the FEM software. Thus, in the first step, with the help of the *Move Components* command, we approach the position where the teeth are almost connected. It is important to do this within the *No Analysis* setting. When we are almost in the position of tooth connection, we turn on the *Physical Operation* option, which simulates that the driver gear (the pinion), in this case the upper gear, drives the driven gear, which means that it becomes the lower gear. The setting can be observed in *Figure 4*.

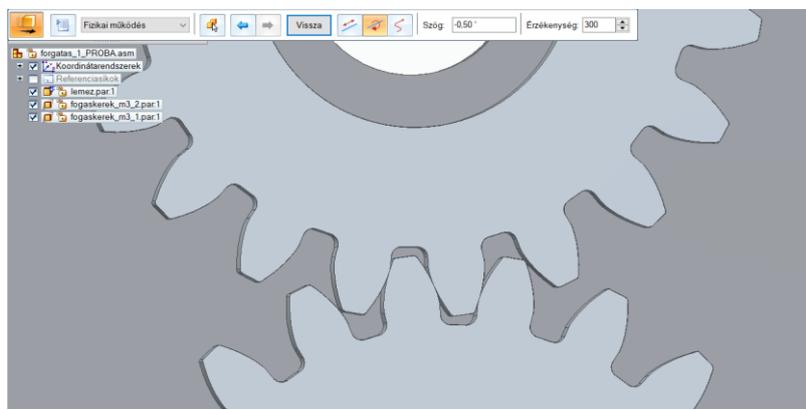


Figure 4. Set up simulation of physical operation

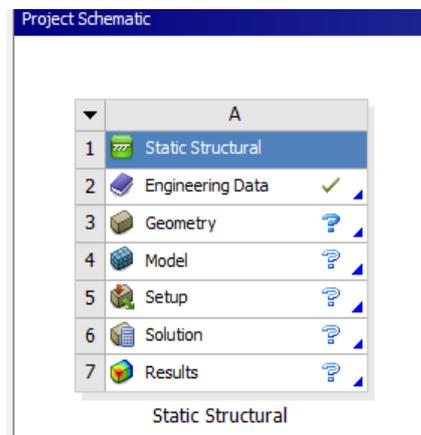
With the help of this command, we perform the rotation every half degree. We save each state separately in an intermediate format. This format is the STEP format, which can also be managed by FEM software. With the help of states saved in the intermediate file format, we can already start running FEM analyses. To do this, we use the free student version of the Ansys software used in the Institute.

4. A FEM TEST FOR GEAR CONTACT

Therefore, to perform the test, we use the R2 version of Ansys 2021. After launching the program, which is Workbench, we need to create what kind of test we want to do. Here we choose static structural and create it. Once created, the test status bar is displayed to see what steps need to be taken to complete the simulation. This can be seen in *Figure 5*. The first step is to create geometry. Here, of course, we read the previously saved STEP format model.

Figure 5

The first step of the FEM investigation



After reading the geometry, double-clicking on the model tab will start the AnsysWB Module. In it we can create the details of the analysis. First of all, the auxiliary plate is turned off from our geometry so that it does not participate in the analysis, since it does not affect the connection of the teeth, so we do not tie up unnecessary computational capacity. You can do this by selecting the disk part on the *Geometry* tab and right-clicking it with *Suppress body* command. This part also disappears from display. After that, meshing should be carried out. On the *Mesh* tab, you can create it by command *Generate mesh*. The finished mesh is shown in *Figure 6*. Here we keep the mesh created by the program, and we go on with it.

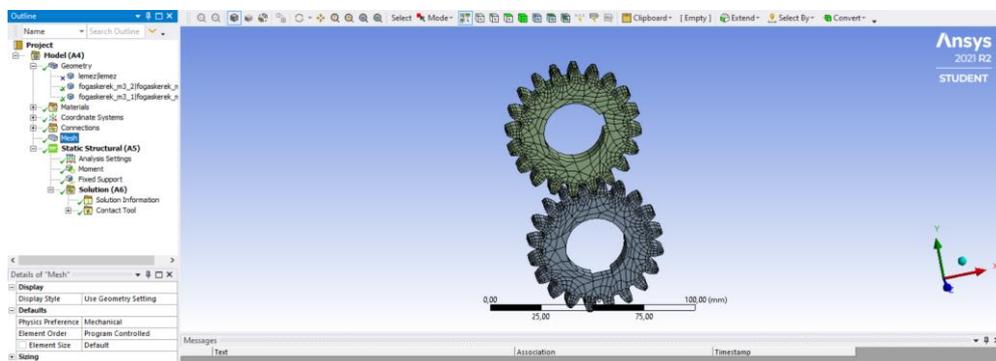


Figure 6. The finished meshing without the auxiliary plate

The next step in the *Static Structural* tab is to determine the various constraints and forces, with one expression, the boundary conditions. In our case, we put a torque on the upper gear, which gives us the drive by selecting the inner surface of the hole and operating the torque of 300 Nm on it. However, in order to create a load on the tooth, the driven gear must be braked. To do this, a constraint called *Fixed support* is used, which is determined on the surface of the hole of the lower gear. This step is shown in *Figure 7*. The torque is marked with an arrow, showing which direction it acts in, and highlighting in red the surface on which it is operated. In blue, it represents the fixed support.

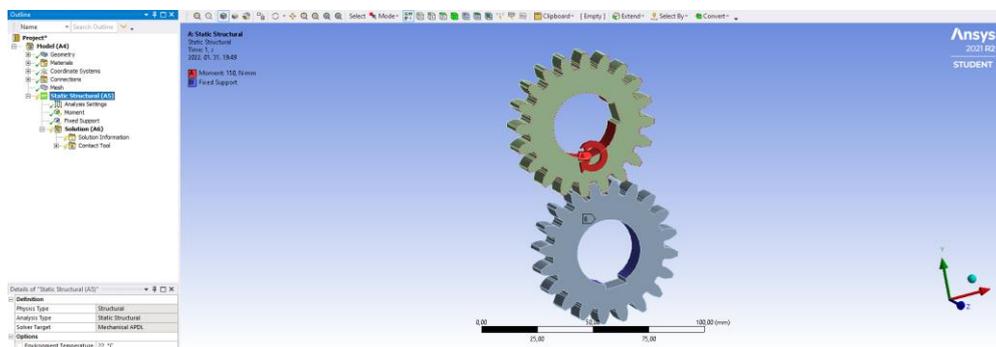


Figure 7. Description of created constraints

After the constraints are determined, the solution can be created. Here, by clicking on the *Solution* tab, we can use the *Contact Tool* to get the results of various connection tests. The most important thing for us now is to see these results. The solution is obtained by running the analysis. Thus, by clicking *Solve*, the results are calculated. The result is shown in *Figure 8*.

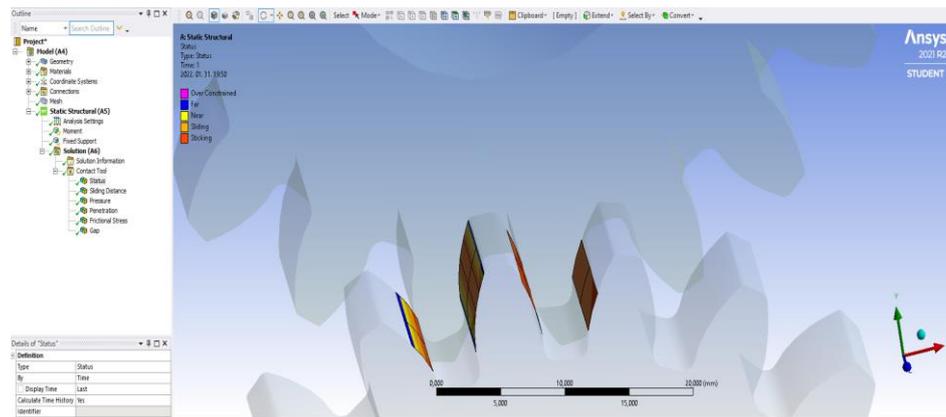


Figure 8. The result obtained for connecting teeth

The result clearly shows the connection between the teeth. For us, the important information is given by the yellow and orange color. This is where the connection is made. The yellow color is the *Near* range, where the teeth are close enough to each other to make the connection. And the bright orange is the *Sliding* range, where the two teeth slide on each other. In essence, it is these two areas that provide the contact pattern image. The analysis shows that there is a striped character that meets our expectations, because the connection with these teeth is formed along a line. However, a result was also achieved that presents a flaw. The connection appears on both sides of the tooth. With the correct gear connection, the connection should only appear on the loaded tooth side. From this result we can say that the calculated center distance is 60 mm, but in practice this is not appropriate, as this creates a connection without a clearance. So, it is necessary to change the center distance in order to create the backlash. Thus, it is necessary to modify the CAD model.

5. MODIFICATION OF THE CAD MODEL USING SHAFTS

Since FEM analysis revealed that the theoretical center distance is not connected to the teeth correctly, we have to go back to the model to make modifications on it. If we return to the CAD model, we will further develop the model to simulate the effect of deformation of the elements associated with the gears with the help of the FEM program. We create two shafts on which we can fit the gears. However, the gears will not go to the center of the shafts, but to the third distance of the shaft. Therefore, we expect the result to be a change in the contact pattern image described in the standard. The modified CAD mounting model can be seen in *Figure 9*.

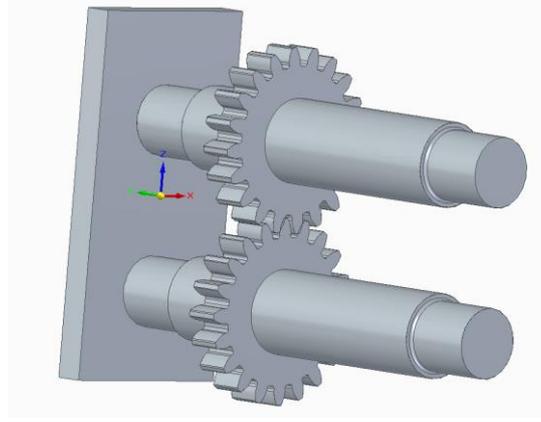


Figure 9. The modified CAD model

In the modified CAD model, it is also possible to rotate during physical operation. It is carried out with the difference that in this case we rotate not the center line of the hole, but along the center line of the shaft. We save the models with a half-degree rotation, in intermediate STEP format, similar to the previous one.

6. FEM TEST OF THE SHAFT-MOUNTED GEARS

The initial steps of the study are similar to the previous case. In the same way, we perform a Static Structural test and read the geometry as described above. The auxiliary plate is also turned off (suppressed) here so that it does not participate in the simulation. The first deviation in settings can be found in the *Connection* tab. In this case, the contact between the two teeth is set to *Frictional* type, which allows us to specify that there is rolling and sliding between the two teeth at the same time, which is also present in a real gear drive. This value is set to $\mu = 0.05$, which between sliding and rolling. The setting you have made is shown in *Figure 10*.

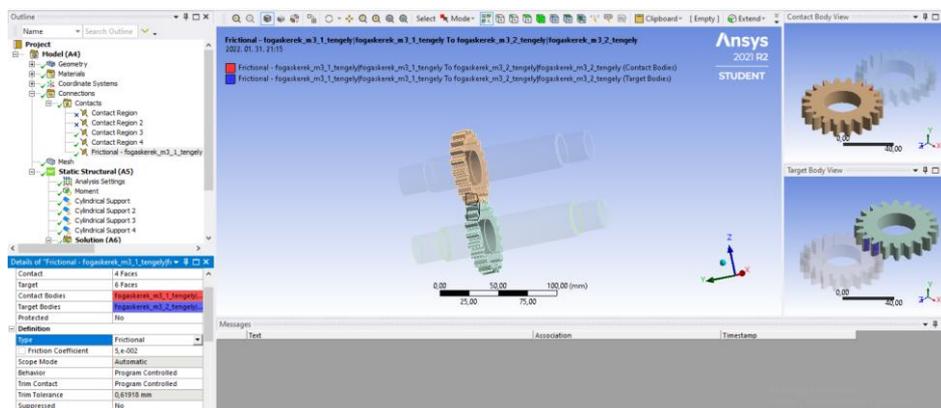


Figure 10. Taking into account the impact of sliding and rolling

The next is to set the constraints on the Static Structural tab. Since we have already created an axis here, we will not put the torque on the gear, but will put it on the shaft. Thus, the torque will be transferred through the shaft from one gear to the other gear. The supports are also different in this model. Here *cylindrical support* is applied to the shaft steps, essentially, where bearings would be located in the case of a gear drive. In the case of the drive shaft, the two supports must be allowed to rotate so that they can drive the driven gear and shaft. Thus, in the settings, in the constraint, it is possible to specify whether the radial, axial, and tangential directions are fixed or free of the type of displacement. In the case of the drive shaft, we allow the tangential direction to rotate, but not in the other two directions. This setting is shown in *Figure 11*.

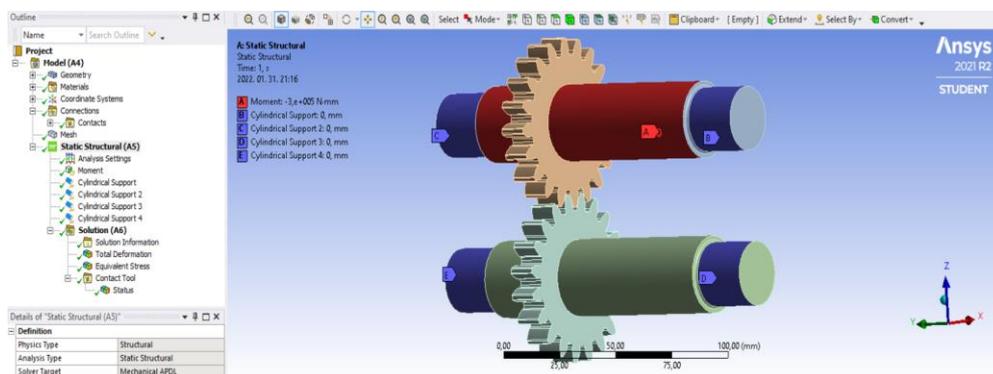


Figure 11. Use of constraints

Now we can run the analysis solution. The solution can be examined in *Figure 12*. It can be seen that, as described in the standard, the contact zone is deformed [6]. On one side appears a more compressed area, formed due to the asymmetric distribution of the load. So the FEM analysis returned the result we expected.

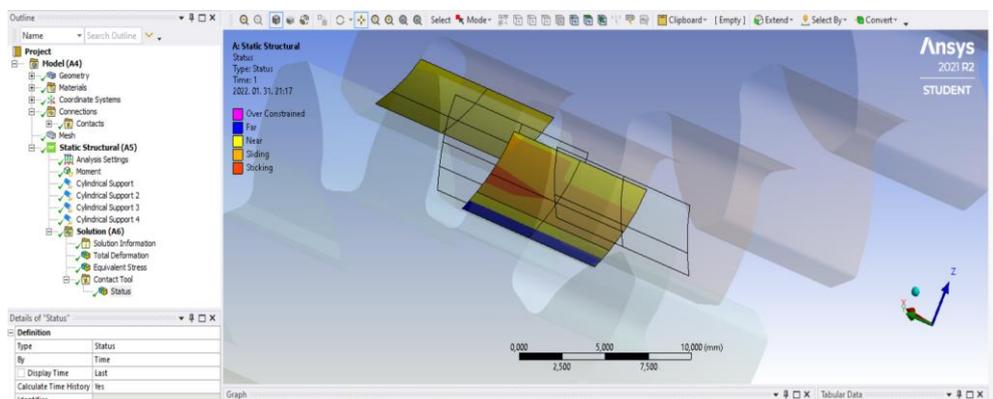


Figure 12. Distortion of the bearing area in case of installation error

7. FORMATION OF THE CONTACT PATTERN BY SUMMING UP SEVERAL ANGULAR POSITIONS

In order to determine the contact pattern and its change, the analysis test presented above must be carried out in several angled situations. By placing these situations next to each other, you can see how the position of the connection line will change depending on the rotation. So we also perform tests on STEP geometries that were previously saved every half degree. These simulations are carried out on a gear connection mounted on the shaft, taken later. The first position is started from a relationship where just one pair of teeth connected. A picture of the initial position is shown in *Figure 13*.

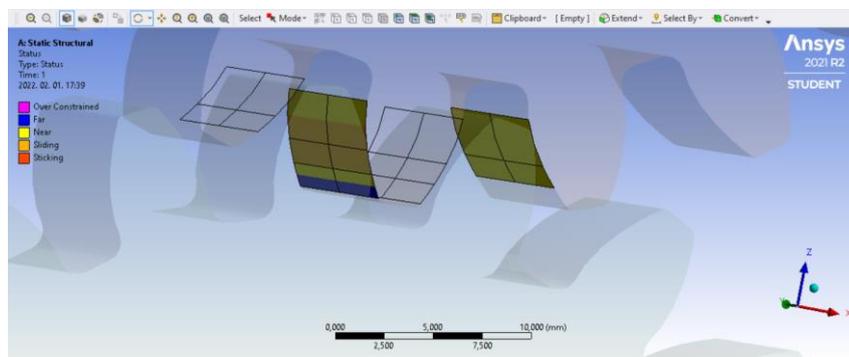


Figure 13. Initial position of meshing-one tooth pair in contact

The next step is to move forward with relationships. However, what we achieve in a position where two teeth connections would already be established, the analysis indicates an error. This is due to the fact that the value of clearance is already less than the element size of the mesh, so the FEM program will detect more connections and will not give us real results. We need to find an opportunity to solve this problem. The solution is to thicken the gear mesh to such a size that the size of the tooth gap does not cause problems. We can do this on the *Mesh* tab, using the *Sizing* command. Here we select our two gears and adjust the size of the mesh to 0.015 mm. This puts us below the value of the tooth gap. A picture of the completed, reduced-size meshing can be seen in *Figure 14*.

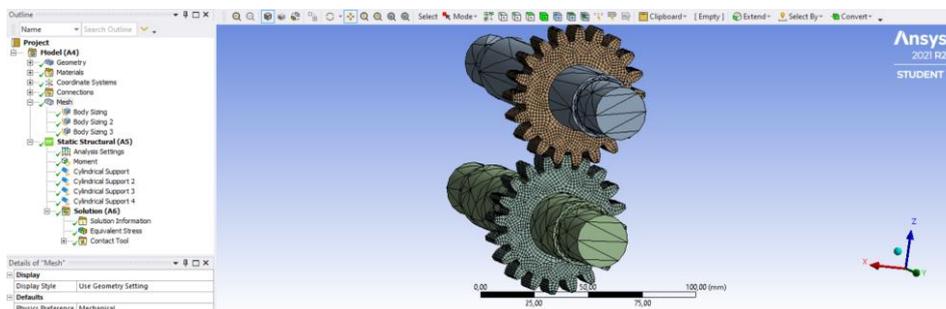


Figure 14. Compressed meshing

Thus, everything is already given to be able to take the contact pattern picture and its change after each turn. In *Figure 15* we can see a change in this displacement, starting from the one tooth relationship, then the entry of the two teeth connections and finally again when only one tooth will be connected. Stress values can also be observed that just when a tooth is in contact, the stress value is distributed on that one tooth, but if two teeth are already connected, the stress value per tooth will decrease.

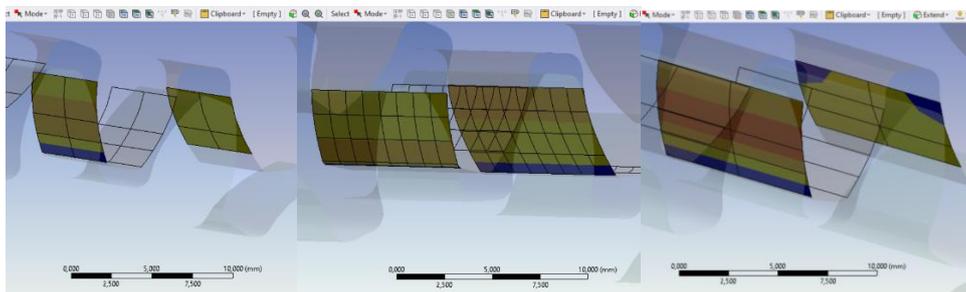


Figure 15. Change of the contact pattern during rotation

8. SUMMARY

In the publication, we prepared the involute section for the theoretical tooth profile and used it to draw the gear. Tests were carried out to see how well the gear generated by CAD systems matched the profile of the theoretically accurate gear. In addition, we have created the installations necessary to be able to perform tests with FEM software. During the study, we created two CAD models that are suitable for physical operation, thus providing the basis for the examination of FEM analysis. During the test, it was determined that the theoretical center distance is inadequate, so its value needs to be corrected. We also conducted a study in which we observed the evolution of the contact zone, if we did not just introduce the gears themselves into the CAD model. The effect of the design on the contact zone, and through this on the contact pattern image, can be detected already during the design phase, and already at this stage we can make observations about how the connection will develop in the case of different designs and what problems they can cause during operation. Contact tests are carried out in several angular positions, and then the resulting contact pattern images are summarized to obtain a picture of the gear.

ACKNOWLEDGEMENTS

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