

MODELLING POSSIBILITIES OF BOLT CONNECTIONS WITH FINITE ELEMENT METHOD

FERENC SARKA

*University of Miskolc, Institute of Machine and Product Design,
H-3515 Miskolc-Egyetemváros
ferenc.sarka@uni-miskolc.hu
<https://orcid.org/0000-0003-3136-4248>*

Abstract: In the article, the possibilities of simulating bolted joints in the Ansys finite element program are presented. The purpose of the publication is to draw attention to the various simulating options. In the article, different simulation options are used for the same model. The versions to be used are 1) bolt connection without preload 2) bolt-tool, 3) beam-tool, 4) real screw thread modelling. In the article, I rely on previous publications, so some stages are presented in less detail than others. During the simulations, I compare the number of elements used, number of nodes, running time, stress and deformation results. The model used is a closed space loaded with internal pressure, which is divided into two halves. A gasket is installed between the two halves, the two halves are fastened to each other by a row of screws. In order to manage the running times, I only examine the part of the entire model corresponding to one screw. The screw joint is loaded with pretension, and the connecting elements with a constant pressure. Where possible in the software, the tightening torque is also taken into account. The results are presented in tabular form for easier comparison.

Keywords: *Ansys, Bolt-tool, Bolt pretension, Beam, simulation of bolt connection*

1. INTRODUCTION

In technical life, we come across many machine elements that basis appeared as far back as ancient times, screws are among them. The creation/description of the helix is linked to the name of the ancient Greek scientist Archimedes. As early as III century BC. he created the mathematical description of the helix, on the basis of that several inventions were born (for example, the water-lifting screw). With the help of a flat cross section guided along a screw line, a screw thread – used since ancient times – can be created and the screw joint created by its application. Today, screw connections have become the most frequently used fasteners, and have even become the most common machine components. They are widely used in the machinery industry due to their simplicity, relatively low prices and easy installation and removal, as well as the possibility of repeating the installation almost any number of times. Unfortunately, the compulsion to cut costs and reduce assembly times results in screw joints being pushed out of more and more places, which is contrary to the aspirations of a sustainable economy that is gaining more and more voice nowadays

and puts less strain on natural resources. Unfortunately, money is always the lord, which is why we constantly destroy our environment (respect to the few exceptions). The exchangeability and repairability of defective elements and thus the extension of the life of products (even vehicles) is more easily achieved by using screw connections. Based on the aspects described above, it is important to have as much detailed knowledge as possible of the ‘functioning’ and simulation of bolted joints.

The finite element simulation of bolted joints was investigated by many researchers (both homeland and foreign) previously, based on many different aspects. It is impossible to summarize the results of all of them in this article, so I do not attempt to complete the literature review. This research work is a continuation of an already started study, but it is also a preliminary study, a research on the optimisation of the layout, tightening torque and tightening sequence. (Lovas, 2018), (Lovas, 2020), (Sarka, 2022), (Bocko, Lengvarský & Huňady, 2020), (Alfattani, 2020).

2. THE USED CAD MODEL

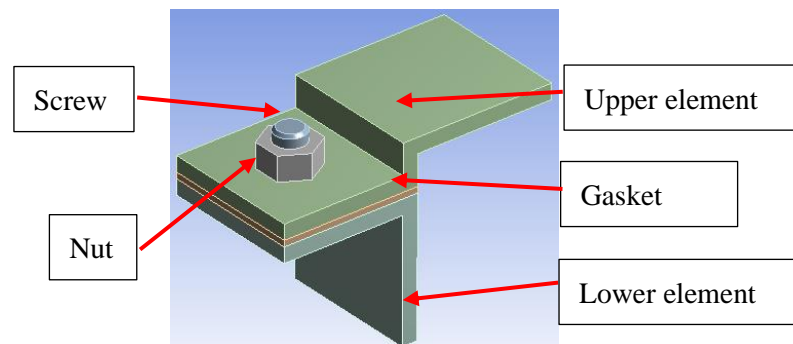


Figure 1. The used CAD model is displayed in the finite element program

In the publication, I examine the part of a container loaded with internal pressure that falls on one screw. The model can be seen in *Figure 1*. I leave out the chamfers, fillets, rounds and the washer from the screw connection, for easier handling. The fasteners are M10 size.

The model consists of a lower and an upper element, which are connected by a screw connection, with a gasket between them. The screw connection is made up of two elements, one is the nut, the other is the screw, neither of them contains a thread. The material of the gasket is silicone, the other elements are structural steel. When creating a finite element model, it is very important to set the contact properties between the elements. By default, the Ansys program sets all contacts to ‘bonded’, which actually behaves as if the two surfaces are glued/welded together. Unfortunately, we have to modify this. The contacts on both sides of the gasket, between the screw head and the plate and between the nut and the plate must be set to ‘rough’, which allows the separation of the elements i.e., only compressive force can be transmitted through the surfaces. To achieve the exact solution, enable the option of large

deformations on the ‘analysis settings’ tab. If we don’t want to create a tight connection, that is, if we put the nut on the screw without tightening it, we are ready. The boundary conditions can be seen in *Figure 2*. The value of the internal pressure was set to 0.2 MPa, and the lateral surfaces were fixed.

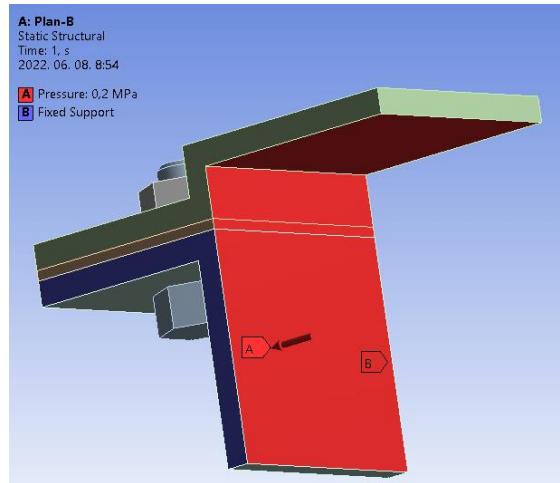


Figure 2. The boundary conditions

This is the simplest model of a screw joint. I did a trial run to see how the conditions were developing. The displacements and stress distribution can be seen in *Figure 3*.

We can retrieve the gasket conditions from the system using the ‘Contact Tool’. With this, we can even check whether the gasket really insulate in place, or whether a leak may occur as a result of a given internal pressure (*Figure 4*). The running time was 180 seconds (main parameters of the computer used: Intel core i5, 5th gen, 16 GB RAM). Of course, in technical practice we rarely use bolted joints without pretension, so in the next simulations we will switch to simulating pretensioned bolted joints.

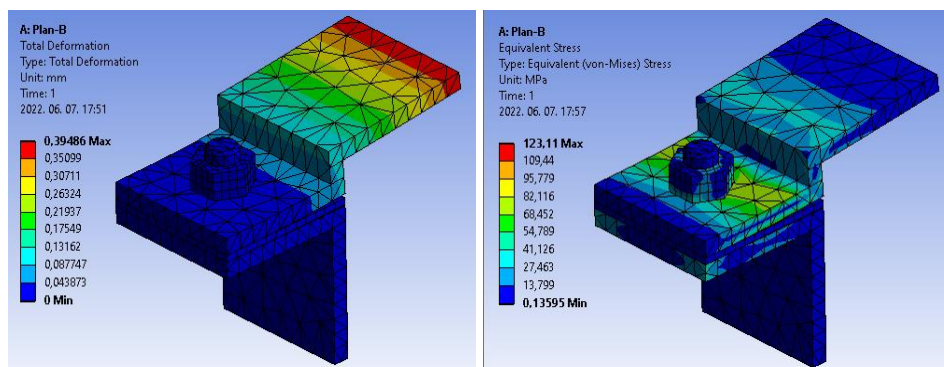


Figure 3. Finite element model of bolted connection, displacements, and stresses

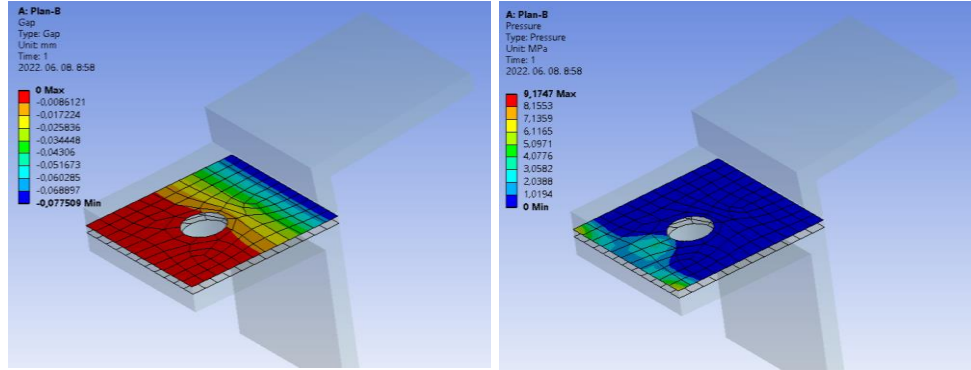


Figure 4. Contact Tool results

3. USING ‘BOLT PRETENSION’ AND ‘BOLT-TOOL’

In the Ansys finite element program, the ‘Bolt pretension’ and ‘Bolt-tool’ options were developed to model the clamping force created by bolted joints. In this case too, it is sufficient to model the screw without thread. The ‘bolt pretension’ is found among the loads, we have to apply it to the cylinder symbolizing the screw shank and enter the value of the pretension force in N. In a real installation case, the screw is tightened/pre-tensioned first, the loads resulting from the operation of the given structure only appear afterwards. We can specify this in the finite element program as well. It can be specified in the ‘Analysis setting’ section, where the number of time steps must be set from 1 to a larger. For the present study, I chose 3. In the first-time step, the screw is pre-tensioned, which pre-tension must remain until the end of the simulation (*Figure 5*, circled in red).



Figure 5. Adjustment of the time course of the screw preload

In the third time step, the internal pressure appears as a load. The value of the load was set to 0.2 MPa, as it was in the previous case. *Figure 6* shows how the value of the internal pressure changes as a function of time.

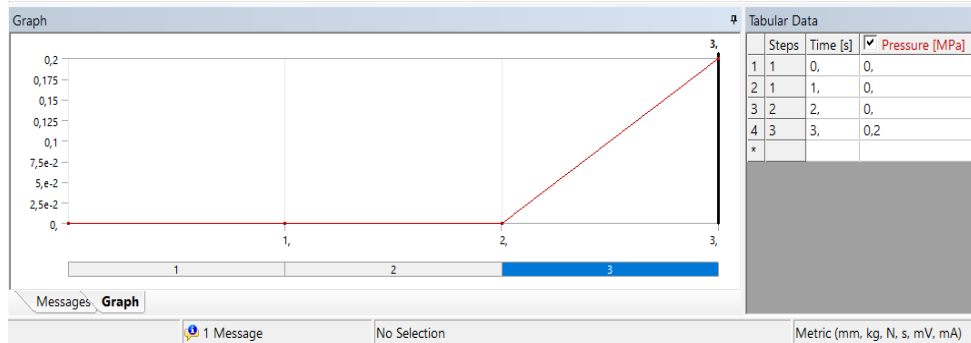


Figure 6. The build-up of internal pressure over time

The results of the simulation can be seen in *Figure 7*. The value of the deformation decreased slightly, and the value of the stress increased. The changes are not significant, 0.04 mm and 7 MPa.

The simulation ran in 240 seconds, which is already a significant increase (33%) compared to the previous test, but it characterizes the real situation much better. The visible change in the results of the ‘contact-tool’ is presented in *Figure 8*, which is already significant compared to the results of *Figure 4*.

Observing the gap result, it can be seen that the completely closed surface part has become larger than it was before, it completely surrounds the screw hole. And the pressure on the seal increased from 9 MPa to 11 MPa, which is an increase of 22.5%. The increase in contact pressure is caused by the deflection of the screw shank.

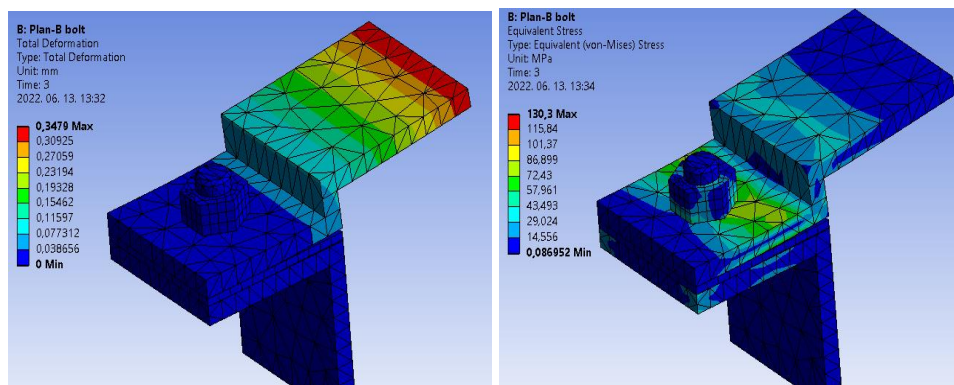


Figure 7. A pretensioned bolt connection simulated using ‘Bolt pretension’. Displacements and stresses

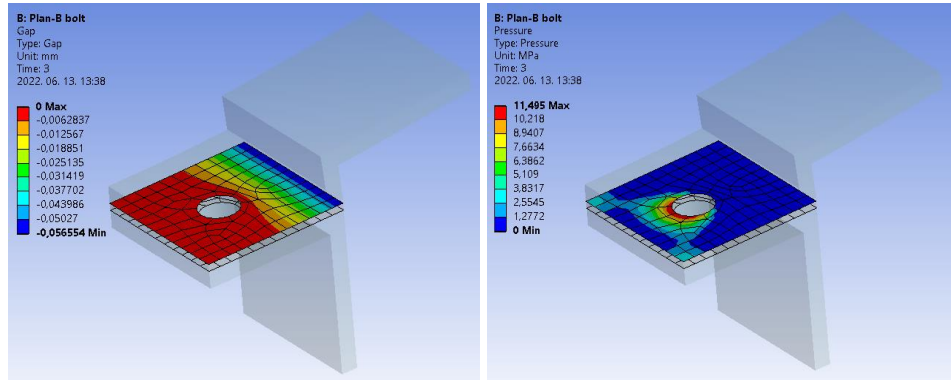


Figure 8. Results of 'Contact Tool' using 'Bolt pretension'

4. THE USE OF THE 'BEAM TOOL'

The ANSYS program gives the opportunity to incorporate various elements into a model already created in a CAD program, such as (spot welding, spring, joint, bearing, or beam). During the simulation I will use the 'Beam' command among the many options. The existing bolt model is removed with the 'Supress' command and a bar is placed in its place with the 'Beam' command. The 'Beam' command can be invoked between contacts with an 'insert' command. We need to specify the shape of the cross section of the bar, which will be circular. Modelling the shank of the M10 screw, we need to specify the radius of the cross section, which is 5 mm. After that, we have to specify which geometries define the newly inserted bar. The first geometry is the element that does not move, the second is the element that moves relative to it. *Figure 9* shows the inserted bar (red arrow). The centre line of the bar will be determined by the centre of the two selected surfaces. The elements simulating the screw head and nut will be the selected surfaces (green arrows).

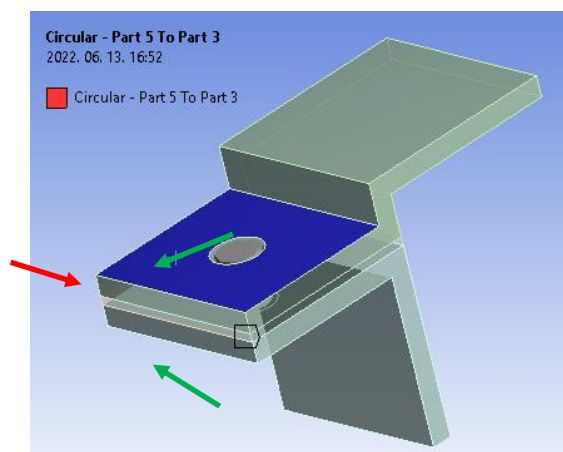


Figure 9. The inserted bar element

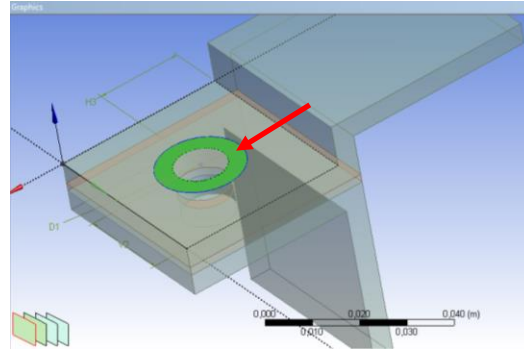


Figure 10. Modelling the bearing surface of the screw head and the nut

The settings described above involve larger surfaces in the analysis than the contact surfaces of the nut and screw head used in the previous case. In order to eliminate this, a surface part must be marked on the previously marked surfaces, which is approximately the same as the contact surfaces of the screw head and the nut. During the simulation, a circle with a diameter of 17 mm is created. This can be done by returning to the CAD system or in the 'Design Modeler' module of the finite element program itself. *Figure 10* shows the model of the contact surface of the screw head marked in green. Of course, this surface must also be created on the bottom plate. During the settings, you have to modify the behaviour of the contacting surfaces. The behaviour must be set to 'deformable' so that the overly stiff surfaces do not affect the results!

After installing the beam as a screw shank, we can see the resulting model in *Figure 11*. All other boundary conditions in the model remained unchanged.

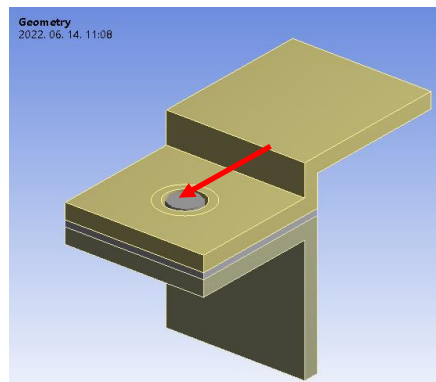


Figure 11. The model used for the 'Beam' tool

In order for the pretension to be applied to the 'Beam' model in our model, we need to drag the created 'Beam' element between the loads, then the 'Bolt pretension' will appear between the loads (*Figure 12*).



Figure 12. Creating a bolt preload when using the 'Beam' tool

The simulation showed the following result after a few seconds of runtime.

We encounter an increase in both displacement (5%) and tension (20%). The increase in maximum stress can be said to be significant, but luckily it is limited to the edge of the hole. The rest of the model falls within the same stress range as the previous simulation. The results of the 'Contact-tool' are shown in Figure 14.

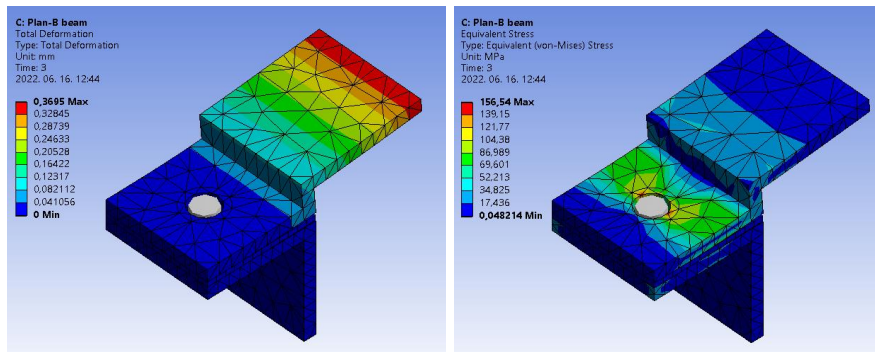


Figure 13. Bolt joint in tension simulated using 'Beam'. Displacements and stresses

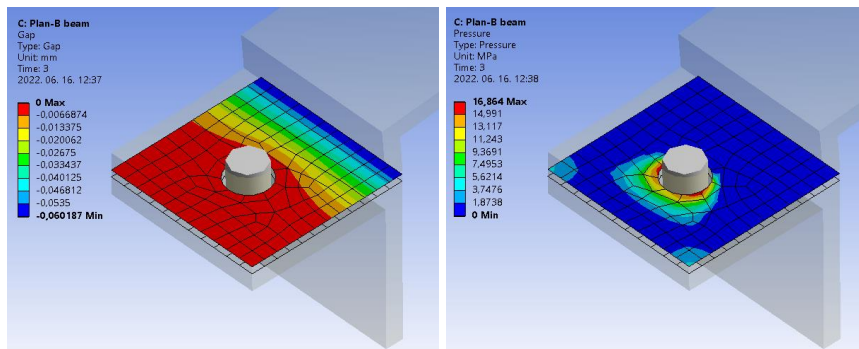


Figure 14. Results of "Contact-tool" using "Beam", gap and surface pressure values

Both the gap (8%) and the surface pressure (48%) increase. The increase in surface pressure is significant. The running time is a few seconds long, which is insignificant compared to the 240 second time of the option presented in Chapter 3.

5. REAL MODELLING OF SCREWS

The closest to reality modelling of screw joints is described in detail in the literature (Sarka, 2022). The results of the simulation based on what was described there are presented below. In the following figure, we can see the applied model and the applied boundary conditions (*Figure 15*). A new load appears for the boundary conditions, and this is the tightening torque of the screw connection. We also meet a new support, which is located on the head of the screw. This support simulates a wrench used when tightening.

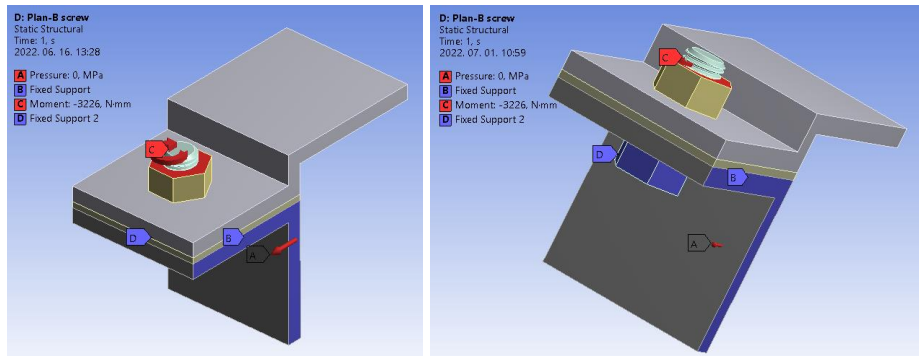


Figure 15. The model, with the boundary conditions

The applied tightening torque was determined based on the literature (Zsáry, 1999), so that the force generated in the screw shaft is the same as the preload applied in previous cases. When setting the contacts, we followed the instructions in (Sarka, 2022). The simulation results are shown in *Figure 16*.

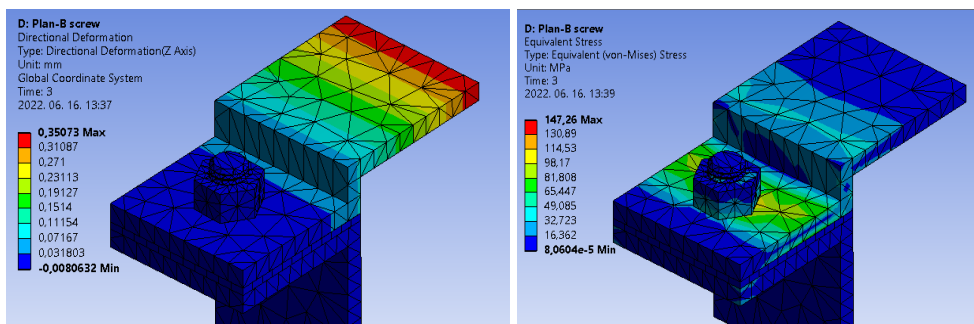


Figure 16. A tension bolt connection simulated using the real bolt geometry. Displacements and stresses

The results obtained from the simulation are close to the results of previous versions. The running time, on the other hand, has increased drastically to a duration of 631 seconds. *Figure 17* shows the gasket conditions in relation to the gap and the contact pressure.

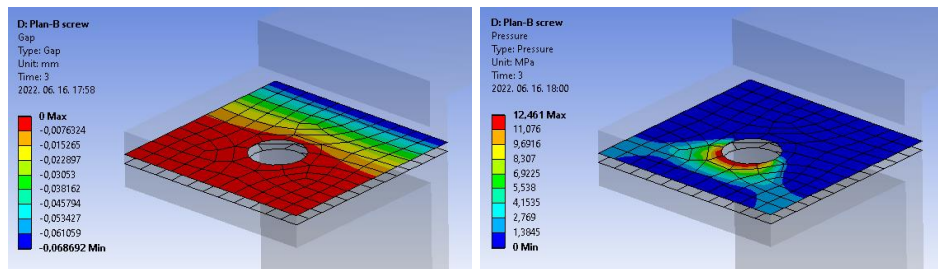


Figure 17. The results of the ‘Contact-tool’ for a screw made with a real thread

In terms of contact, similar values are obtained as with the application of “Bolt-pre-tension”.

6. SUMMARY

The results of the four different cases are presented in tabular form so that they can be better compared with each other. From the four simulations carried out, it can be concluded that there are significant differences between the results of the individual models. If we want to carry out preliminary simulations to help planning, ‘Beam’ is used due to its suitable and very fast running, although the results are not very accurate, but fortunately the deviation points in the direction of safety.

*Table 1
Summary of run results. The highest value is bold marked*

	Def. [mm]	Stress [MPa]	Gap [mm]	p _{cont} [MPa]	Time [sec]	Elements [pcs]	Nods [pcs]
Without pretension	0,41	130,24	-0,08	8,89	4	1638	4935
With pretension	0,35	130,30	-0,06	11,50	240	1638	4935
Beam tool + pretension	0,36	156,54	-0,06	16,86	4	1239	3547
Real screw model	0,36	147,26	-0,07	12,46	631	10010	19108

If we want to get a more accurate picture of the bolt connection, then either the ‘Bolt-pre-tension’ or the real bolt model must be used. Of these, the ‘bolt tool’ should be used if we do not want/need to take into account the screw torque from the tightening torque. In all other cases, ‘real bolt model’ is the right choice.

We plan to compare the revealed results with real experiments and publish them in the future, so that the settings of the finite element method can be even closer to reality.

ACKNOWLEDGEMENT

In this way, I would like to thank Attila Potyka departmental engineer and Zsolt Tóbis master's teacher for the help with preparing the article. They took my work forward with their constructive criticism.

REFERENCES

- [1] Alfattani, R. (2020). Modeling and Simulating the Static Structural Response and Lift Off of a Preloaded Bolted Joint on a Flange Vol. 63, *Proceedings of The 14th International Conference on Interdisciplinarity in Engineering*. Târgu Mureş, Romania: MDPI.
<https://doi.org/10.3390/proceedings2020063010>
- [2] Bocko, J., Lengvaský, P. & Huňady, R. (2020). Possibilities of modelling the bolts in program ANSYS. *IOP Conf. Series: Materials Science and Engineering*, IOP Publishing.
<https://doi.org/10.1088/1757-899X/776/1/012021>
- [3] Lovas, L. (2018). Csavarkötés modellezésének kérdései. *GÉP*, 69 (4), 59–62.
- [4] Lovas, L. (2020). Haszonjármű kerékcsavar lazulásának kérdései. *GÉP*, 71 (7–8), 43–46.
- [5] Sarka, F. (2022). Examination of Bolt Connection with Finite Element Method. In Jármái, K. & Cservenák, Á. (ed.), *Springer Cham., Vehicle and Automotive Engineering*, 4, 212–222.
https://doi.org/10.1007/978-3-031-15211-5_19
- [6] Zsáry, Á. (1999). *Gépelemek I.* Budapest, Nemzeti Tankönyvkiadó.