Finite Element Simulation of the Rupture of Tendons

Dénes Faragó[1,2](✉), Dániel Takács[1], and Rita Mária Kiss[1]

[1] Faculty of Mechanical Engineering, Department of Mechatronics, Budapest University of Technology and Economics, Budapest, Hungary
farago@mogi.bme.hu

[2] Faculty of Mechanical Engineering, Department of Mechatronics, Biomechanical Research Centre, Budapest University of Technology and Economics, Budapest, Hungary

Abstract. In the case of the transplant surgery of ligaments, it is common practice to use tissues from other parts of the body with similar properties. Although they have similarities, there are some distinct properties, some of which may have serious effects on the success of the operation. These properties can be measured and compared to each other so that the surgeon can choose the best option.

The goal of the present research is to validate the measured data of the tensile tests of tendons and the theoretical results on fibrous materials, with special regard to the rupture phase. The mechanical properties of the tendon vary significantly, so in the case of a static tensile test, the material's rupture is not instantaneous, but consisting of multiple steps, as the different groups of fibres are tearing. The main objective is to model this behavior and to compare it with previous results.

Our model is based on the results of in-vitro measurements and fibre bundle models describing the behavior of compound fibrous materials. Then by using finite element simulation, the realistic process of the rupture is approximated by a bilinear debonding process. The fibres' mechanical properties can be set with distribution functions, representing the real variance of the tissue. In the method used, the tensile strength of the fibres is represented by contact stiffness, like static friction, practically separating the fibrous body examined into two parts. With the proper modelling of the fibrous structure's stochastic behavior, it is possible to validate the previous in-vitro results. The method can be used in other areas, such as the examination of ligaments possessing similar properties, or in the description of the behavior of fibre-reinforced composites.

Keywords: Finite element simulation · Ansys · Tendon · Static and dynamic test · Mechanical properties · Validation of results

1 Introduction

Exercise is an integral part of human lifestyle, for which the proper and healthy functioning of the musculoskeletal system is essential. In case of damage, knowledge of system behavior is required to restore the system to its original state.

T. Jarm et al. (Eds.): EMBEC 2020, IFMBE Proceedings 80, pp. 553–562, 2021.
https://doi.org/10.1007/978-3-030-64610-3_63

The musculoskeletal system is made up of several different components, each with a specific function. According to one of the simplest groupings, the system consists of active and passive elements. Typical passive elements include the bones and cartilage, which form the rigid frame of the whole system, and their primary function is load bearing. Muscles that are capable of exerting force, thereby allowing the system to move, are considered to be active elements [1, 2].

There are several critical elements of the musculoskeletal system that cannot be so easily classified. These include tendons and ligaments. The function of the tendons is to connect the muscles with the bones, i.e., to transfer the power exerted by the muscles to the rigid frame.

People's active lifestyles often lead to minor or major injuries to the musculoskeletal system. The regeneration of tendons and ligaments is very slow and to a small degree, and in practice, in severe cases, transplantation is the preferred solution [3–6].

Therefore, most of current biomechanical research is on the mechanical examination, sterilization, and storage of tendons. They are intended to assist doctors from the engineering sie by measurements and theories.

Validation is needed to support these measurements and the theories derived from them. One way to do this is to run finite element simulations, the results of which can confirm assumptions. Besides, the method reveals what measurement results can be expected.

The study aims to develop a simulation method that, when compared to tensile results, gives a satisfactory approximation to reality. It is essential that the resulting arrangement is as simple and easy to use as possible, and that it requires small computing capacity to process large amounts of data [1]; [3–6].

2 Method

We used the finite element software of ANSYS 19.2 Workbench student version to develop the simulation method. We used the results of the tensile testing of several types of tendons that were subjected to mechanical testing, and their mechanical properties consequent upon different sterilization methods were published [7, 8]. The results of the tensile test and the tear curve were available for each tendon.

It makes things difficult that the material under investigation has a non-linear behavior since collagen fibers have viscoelastic properties, and the fiber arrangement is not parallel and does not have the same initial tension. The inhomogeneity resulting from a structure similar to a fiber-reinforced polymer composite also increases the complexity of the required model. The most significant difficulty with the simulation is that the material has to be tested until it has completely failed. For these nonlinearities, there are several treatment methods in the Ansys software. However, simulating material splitting is a big problem because the other difficulties are only distorting the mesh generated. Still, here you need to model the full splitting to get real data.

In the version used, only a limited number of nodes and finite elements can be used, so the model needs to be simplified.

The measurement can also be approached from the theoretical point of view. The method is based on the simplest model of a bundle of fibers, which is a nonlinear, viscoelastic model of the Standard Solid type. It is possible to fit other types of models (for example, a hyperelastic model), but this is not addressed in this study. The strain-stress relationship is as follows (1):

$$\sigma(\varepsilon) = c \cdot \varepsilon + a \cdot \left(1 - e^{(-b \cdot \varepsilon)}\right) \tag{1}$$

This applies to the initial flexible phase before failure. In order for the correlation to be specified for the entire period, it is necessary to multiply the correlation by a confidence function. In our case, this can be derived from a normal distribution with the expected value m and standard deviation S. This is the part of the correlation that represents the tensile elongation distribution of the fibers so that rupture is expected at elongation m. After fitting the parameters required for the current measurement, the final result is shown in Fig. 1.

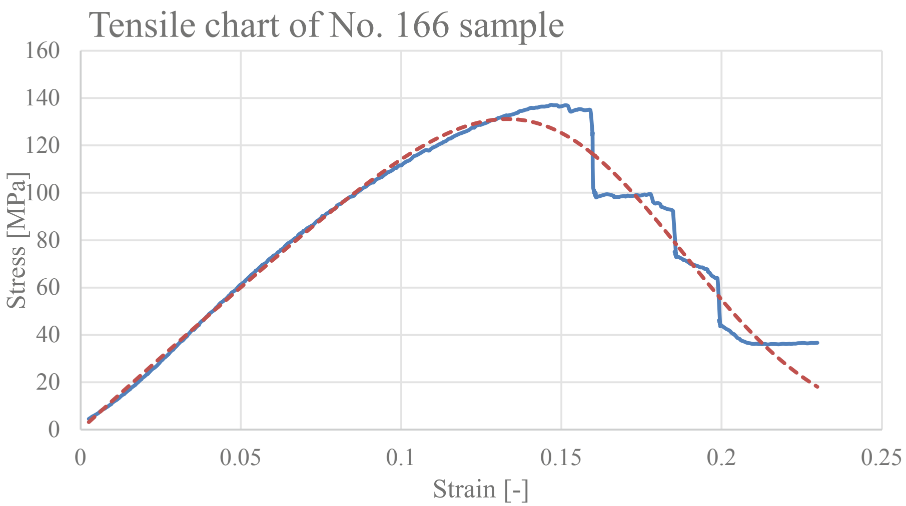

Fig. 1. No. 166 sample measurement and theory comprasion

Figure 1 shows that the method closely approximates the measurement result. By increasing the complexity of the theoretical model (using multiple bundles), the approximation can be improved, or even the cascading character may be regained.

One of the critical tasks of simulation and its problem is to model the rupture. To solve this problem, we have investigated several possible methods, and we refined our method upon examining and testing them.

Out of the possible methods, we used the so-called separation method. This is similar to the phenomenon of adhesion friction. In the case of adhesion friction, the two bodies are stuck together until a critical force is reached, above which the two bodies are displaced relative to one another. This is also similar to the separation method, in which two bodies are joined by some contact, which disappears when some conditions are met.

Any geometry that is connected can be used. This allows the application of a simplified model that has a small number of elements and is easy to operate.

The method is called Cohesive Zone Method Contact Debondig [9]. The technique can also be used for simple Static Structural analysis. The starting point for the study is a two-body model that is bonded to each other by a Bonded connection. After that, it is necessary to add the failure module from which to select Contact Debonding, as shown in Fig. 2.

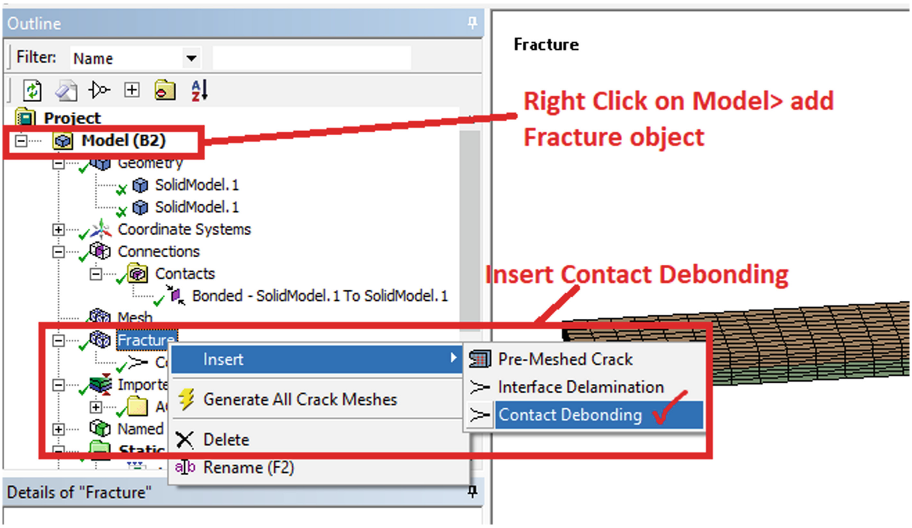

Fig. 2. Introduction of the method in Ansys [9]

2.1 Fiber Separation

Contact Debonding geometry, contact, and material model were created, and due to the limitations of the program, square cross-section geometry was used in our investigations. The two square columns were fixed to each other with Bonded constraints in the center, with Fixed support at one end and Displacement at the other end. For better convergence, several Load Steps were applied.

The fiber separation thus created can be used for realizing multiple fiber separation by multiplying the geometry. The relative arrangement of the cross-sections does not affect the result.

The next step is to create material models for the fibers. In this study, the previously used material model was applied, which has different parameters for different fibers. The more non-linearity is introduced into the model, the harder it will be to find convergence, so everything except Contact Debonding is assumed to be linear. The stochastic behavior of the material parameters of the fibers was modeled so that the settings of the cohesive zone for each fiber contact are the same, so the fibers have a fixed tensile strength. In the system studied, the only difference is in the modulus of elasticity; these parameter values follow a distribution. By specifying the boundary conditions, as in the case of a single fiber, one end of the fibers is trapped, and the other is loaded with a displacement in a plane. As a result, the fibers reach their maximum stress at different strain values during loading, so that they break at different moments. In this case, too, a properly convergent solution was obtained, which adequately showed the expected behavior, as can be observed in Fig. 3. It can be seen that the fibers are broken one by one according to their stiffness.

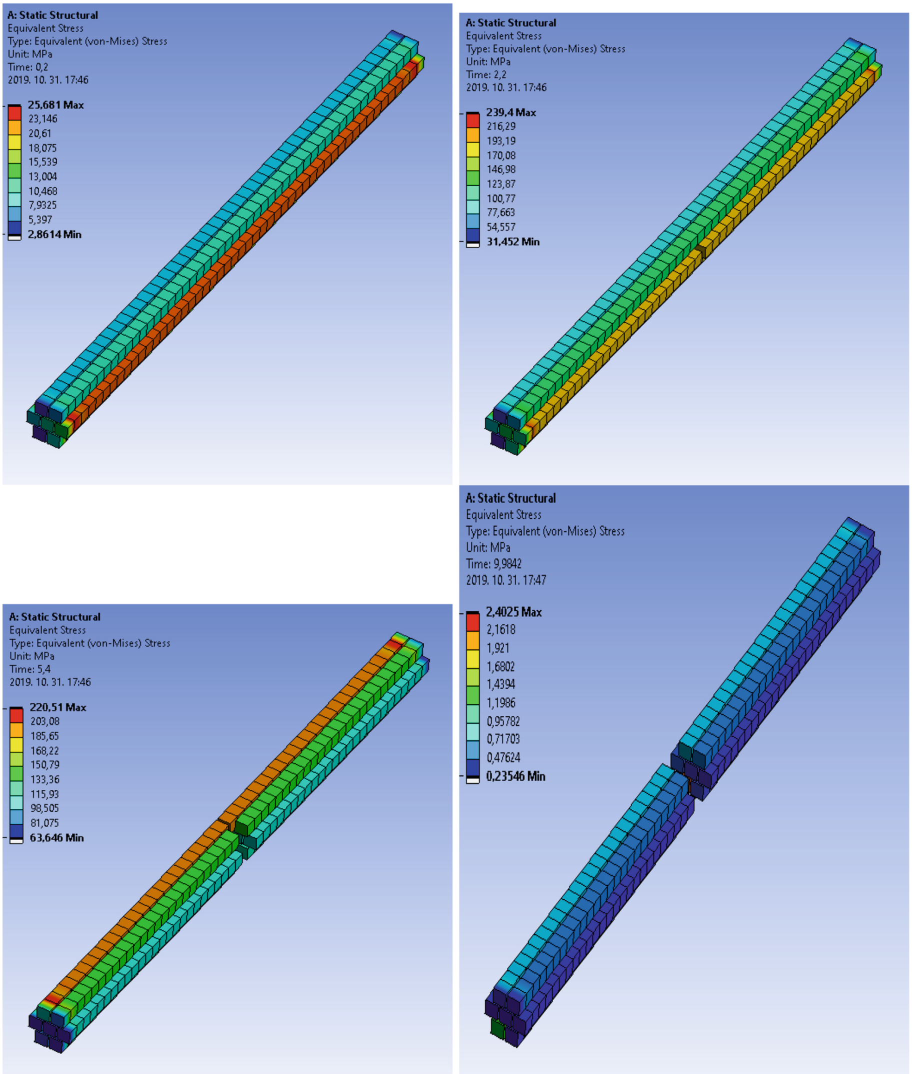

Fig. 3. Stochastic behavior of the fibers

2.2 Preparation of Real Parameters

After the method has been developed, it is necessary to provide real data from the measurement results instead of random input parameters. The measurement and the theoretical model prepared for it are both non-linear and viscoelastic, so the linearly elastic model used approximates the real stress-strain curve only to a certain extent.

The geometry was determined from the results of the measurement before the mechanical test. In addition to the element limit, the maximum number of bodies that can be tested in one assay is 50. Therefore, the fiber bundle was modeled with 25 pairs of bodies.

For each fiber, a different linearly elastic material model has to be created, for which we have given the required elastic modulus. This was obtained from the slope of the straight line fitted to the measured data before the failure. The standard deviation of the elastic modulus was determined by proportioning the theoretically calculated standard deviation. The Poisson factor is 0.5.

The maximum stress was obtained from the measurement results. The separation distance and the value of the artificial damping factor were chosen based by experience.

Data entry. Due to the large amount of data and iteration, it is necessary to parameterize the process. The cross-sectional area or the length can specify the fiber geometry. In this method, the geometry data for additional fibers are automatically generated after entering this data.

The software cannot interpret the specific values of the elastic modulus in this form due to its stochastic nature. Therefore, MatLab is used to generate random values with the specified parameter distribution, which can be read by Ansys. Then a script file can be generated, which is responsible for retrieving the material parameters and creating the material models, to be used to further speed up the simulation preparation process.

Limitation. Fiber separation introduces a high degree of non-linearity into the system. This may result in a lack of convergence needed for the solution, so the software cannot solve the problem. It is possible to manually adjust the contact stiffness (Contact Stiffness) in the Bonded contact settings, which often helps to solve the problem. Thus, this method is relatively easy to apply to model a rupture, but its limitations are worth considering when using it.

3 Results

3.1 Preliminary Results

The method described in the previous section was used. To avoid the Debonding problem, the simulation was run with three stiff contact stiffness values relatively remote from each other (400 N/m, 450 N/m, 500 N/m), each of which already produced convergent results. Afterwards - out of the results obtained from the evaluation of the simulation - the average DC voltage is plotted, which approximates the engineering stress quite well. The result of this method is shown in Fig. 4.

Figure 4 shows that the curve for stiffness 450 N/m is the closest to the curve for the measurement. The staggering of the graph is observed due to the use of small

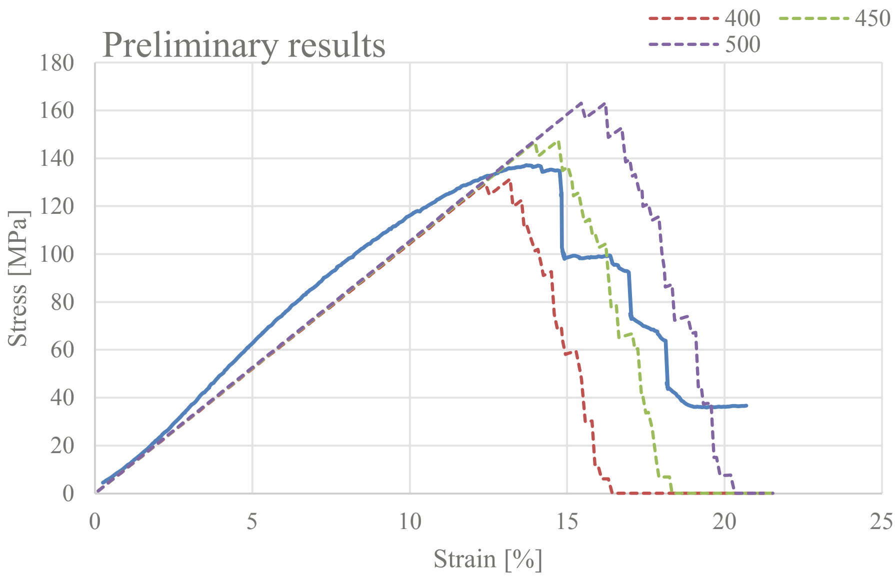

Fig. 4. Preliminary results

amounts of fiber so that their distributional values are quite well separated. This effect can be reduced by increasing the number of fibers.

Using iteration, a stiffness value of 460 is yielded, which approximates the tensile curve obtained from the measurement quite well. The simulation parameters were used to perform additional runs, thus excluding results by only random generation. Further simulations were performed with different data and with a corresponding contact stiffness of 460. Before each simulation, the material parameters were re-generated and then re-analyzed, proving that the method is stochastic.

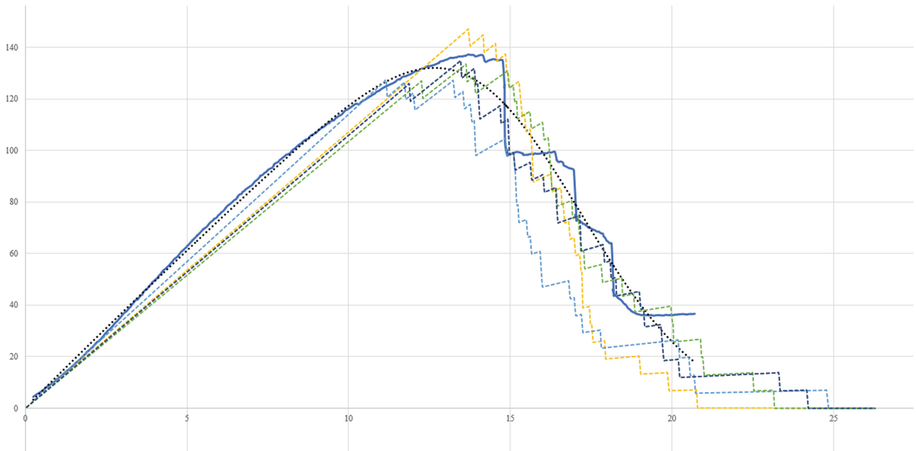

Fig. 5. Stress-strain diagram of results, with different material parameters (For a stiffness value of 460 N/m)

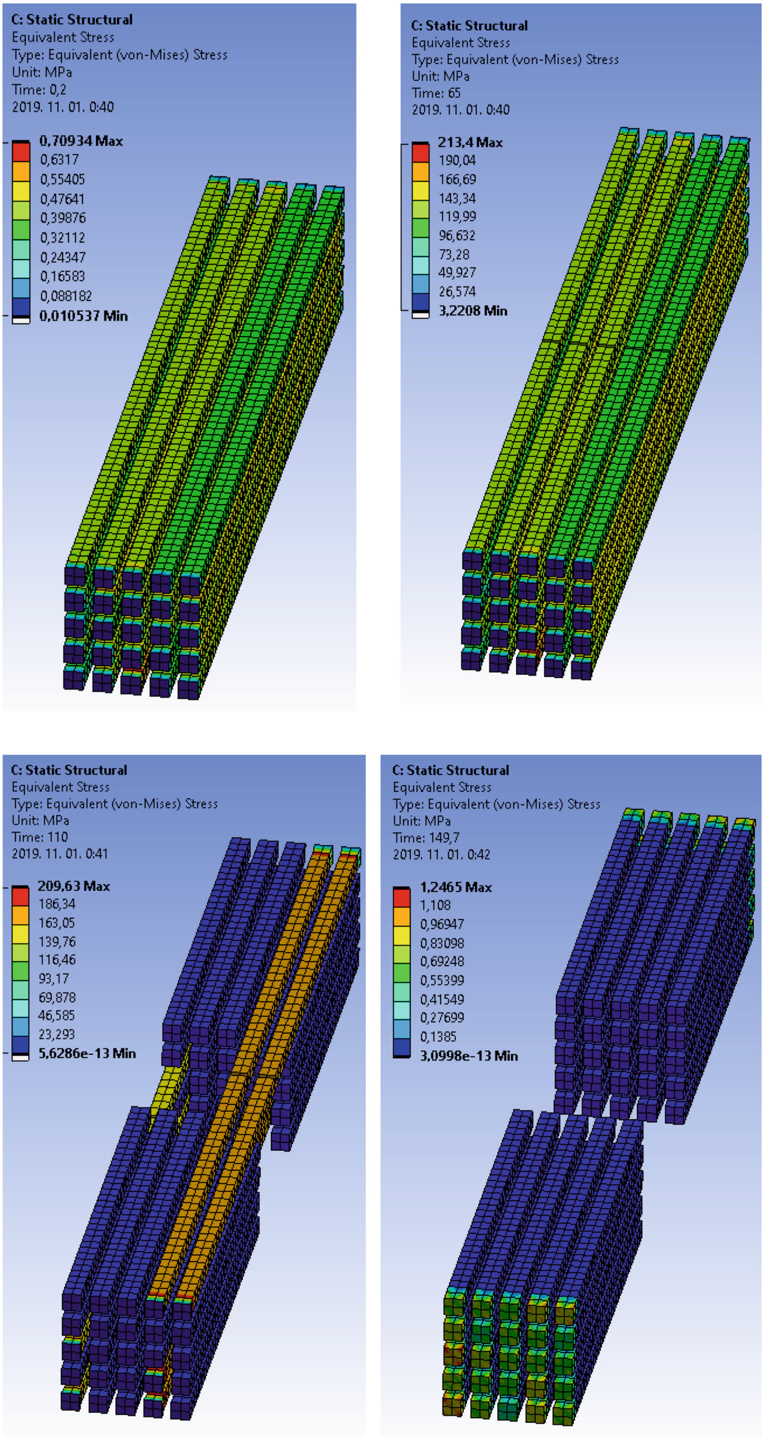

Fig. 6. Simulation of system rupture

The graph of the results is shown in Fig. 5. It can be observed that most of the results obtained are close to the desired curves, and stochastic behavior can also be detected. A simulation of a system rupture is shown in Fig. 6.

4 Conclusion and Discussion

The aim of our study was to develop a simulation method that provides a very close approximation of tendon tests by a tearing apparatus. The method developed is simple and easy to use, requires low computing capacity, and is therefore suitable for processing large amounts of data.

Geometric and mechanical data recorded during the measurement were used for this method. When designing the geometry, the limitations of the software are required to be taken into consideration. Afterwards, the data set from the measurement can be used to determine the parameters to chracterize the material model. After the material models are generated and loaded into the software, the simulation can be run. After specifying the necessary boundary conditions and after defining the failure mode, the simulation can be run.

After evaluating the data, further refinement of the model is possible. This can be done by specific iteration steps, thus approaching the initial measurement or theory. Based on the results, the original objective was achieved.

The method described is further developed to provide a better approximation, and the characteristics of a tendon in a live organism can be predicted without mechanical examination. The method should also be extended to a hyperelastic material model while maintaining the stochastic property.

Acknowledgements. The research reported in this paper was supported by the Higher Education Excellence Program of the Ministry of Human Capacities in the framework of the Biotechnology research area of Budapest University of Technology and Economics (BME FIKP-BIO). This research was supported by the National Research, Development and Innovation Office (OTKA K 116189).

This research was supported by the ÚNKP-18-3 New National Excellence Program of the Ministry of Human Capacities of Hungary under grant no. ÚNKP-18-3-I-BME-183.

References

1. Vas, L.M., Tamás, P.: Fiber-bundle-cells method and its application to modeling fibrous structures. In: 5th Conference on Mechanical Engineering Budapest (2006)
2. Holzapfel, G.A.: Biomechanincs of Soft Tissue, Graz University of Technology. pp. 1–5 (2000)
3. Bojtár, I.: Mechanikai anyagmodellek, Budapesti Műszaki és Gazdaságtudományi Egyetem (2017)
4. Rackl, M.: Curve Fitting for Ogden, Yeoh and Polynomial Models. Technische Universität München (2017)
5. Szakály, F.: Numerical modelling of natural and artificial human ligaments. Budapest University of Technology and Economics (2013)

6. Vas, L.M., et al.: Novel evaluation method of acoustic emission data based on statistical fiber bundle cells. J. Compos. Mater. **53**(17), 2429–2446 (2018)
7. Gyorgy, H., et al.: Does a different dose of gamma irradiation have the same effect on five different types of tendon allografts?—a biomechanical study. Int. Ortho. **41**(2), 357–365 (2016)
8. Gyorgy, H., et al.: Pitfalls during biomechanical testing—evaluation of different fixation methods for measuring tendons endurance properties. Acta Physiologica Hungarica, ***103***(1), 86–93 (2016)
9. https://caeai.com/sites/default/files/CAE_Fatigue_and_Fracture_Seminar_-_CZM_For_Web.pdf. Accessed 15 Jan 2020