Design of Machines and Structures, Vol. 12, No. 2 (2022), pp. 119–129.
https://doi.org/10.32972/dms.2022.023

MECHANICAL SIMULATION OF SPRING-BASED TUBE COMPENSATOR

LAURA TRAUTMANN[1] – ATTILA PIROS[2] – KÁROLY SZILÁGYI[3] – BALÁZS KOMÁROMI[4]

[1]*Budapest University of Technology and Economics,*
Department of Machine and Product Design
H-1111 Budapest, Műegyetem rkp. 3.
[2]*John von Neumann University,*
Department of Innovative Vehicles and Materials
H-6000 Kecskemét, Izsáki út 10.
[3]*C3D Engineering Consultant Ltd.*
H-1106 Budapest, Fehér út 10.,
[4]*ESZO Kft.*
H-2451 Ercsi, Öreghegyi út 2867 hrsz.
[1]*trautmann.laura@gt3.bme.hu*, [2]*piros.attila@gamf.uni-neumann.hu*
[3]*szilagyi.karoly@c3d.hu*, [4]*komaromib@eszokft.hu*
[1]https://orcid.org/0000-0001-9230-4949, [2]https://orcid.org/0000-0002-0829-8062

Abstract: This article deals with the examination of a new product family of tube holders. FEA analysis and analytical calculation also provided information on displacements due to force. The article introduces a simplified mathematical model to substitute the complicated numeric simulations. The results have shown that appropriate calculations are accurate and can replace the simulation of fully detailed model.

Keywords: *Tube Compensator, FEA simulation, Displacement*

1. Introduction

The project described in the article was prepared for the support of ESZO Kft. ESZO Szerelőipari Kereskedelmi és Szolgáltató Kft. was founded in 2006 as a limited liability company. The purpose and activity of the Company are to provide technological tube, steel structure, and machine assembly services for the chemical, oil, energy, and food industry sectors by employing free, valuable, and experienced labour in the Central Hungary region. This information is based on the Product Information Brochure of ESZO Kft. from 2020.

ESZO Kft. has developed its products: spring tube holders. It contains four tube types and nine assemblies of them. There were previous simulations for the tube support and analytical strength calculations of the spring since the spring is essential in these products. It performs the compensation if there is a displacement in the tubing. These calculations gave information about the geometry and the stiffness of the spring; thus, it is known how it will react to the different displacements and with what force it will respond.

The spring is only one part of the product, and there were no calculations for the other parts, however, these are also very informative. Besides these, the available report did not explain the contact goals and places, and the model distribution and the result are also insufficient.

The main question in this research is, how much does the tube holder deform under a specific load? This information about the compensators can help the designers.

The parts of this study were:

– tube holder strength simulation in full detail,
– simplification (geometric transformations),
– simplified analytical calculations,
– design guide.

In order to find these deformities, Finite Element Analysis (FEA) was used, which is usually "used to find stress distribution for complex geometries" [1].

2. METHODS

The compensators are designed to compensate for vertical displacements caused by thermal expansion [2]. Here they can be divided into two main groups: five drawn and four compressed systems.

As a first step, the obtained Solidworks CAD models were converted to STP format by using PTC Creo7, and then the (.asm) Creo assemblies were created in PTC Creo6 *(Figure 1)*.

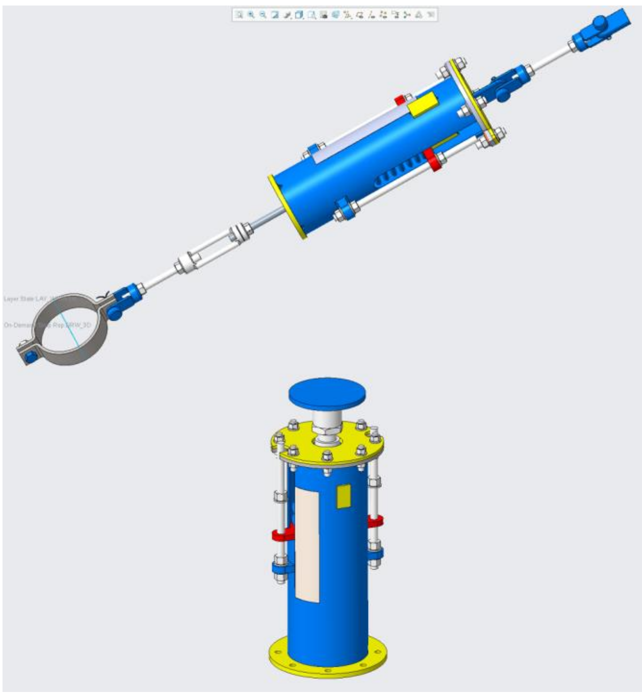

Figure 1. *Fully detailed geometries in PTC Creo 7*

The geometric threads were removed from the threaded parts, and cylinder/bore shapes were created with the same nominal diameter.

An assembly state (Simp Rep [3]) optimized for finite element analysis was created, in which the unnecessary parts were excluded in the force flow, and formal simplifications were performed on the remaining parts (removal of chamfers, roundings, etc.) *(Figure 2)*.

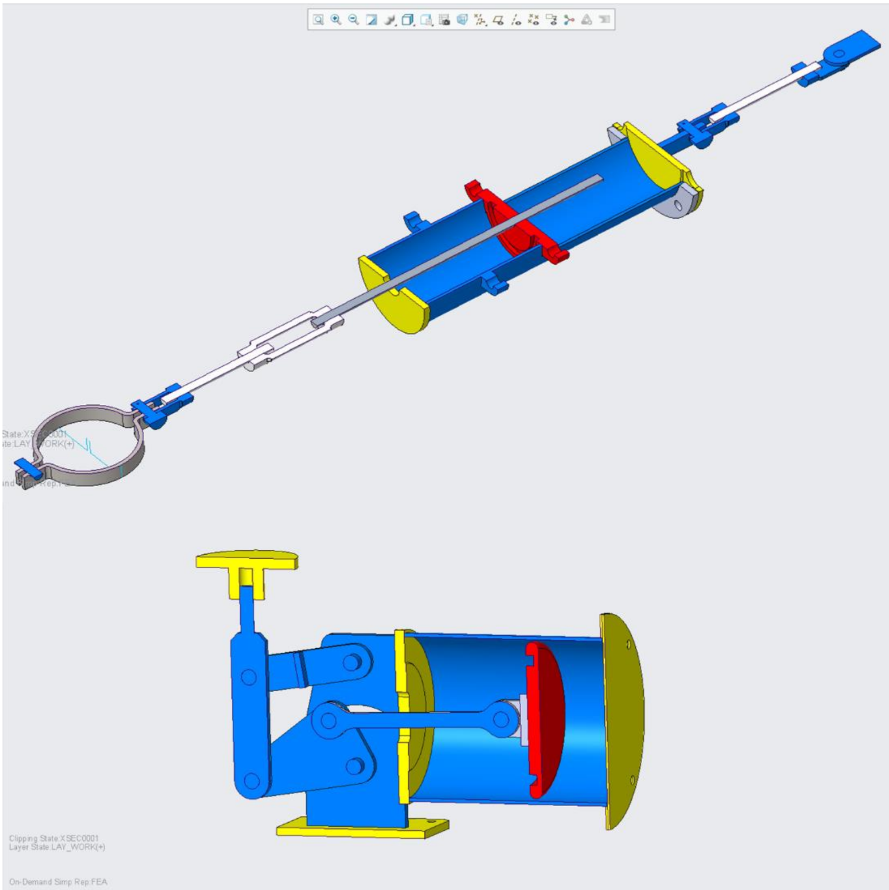

Figure 2. *Simplified geometries in PTC Creo 7*

The compensators can be divided into two subassemblies, between which the spring transfers the load. The subassemblies can be examined separately, thus the assembly states optimized for finite element examination have been created for them as well *(Figure 3)*.

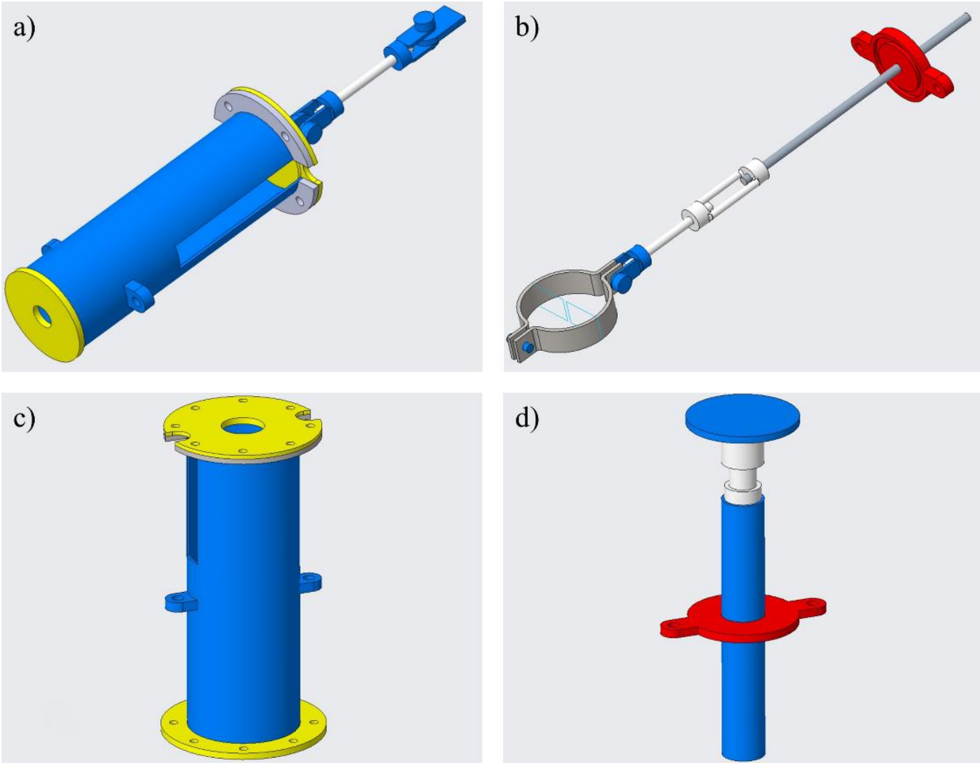

Figure 3. *Separated subassemblies for FEA simulation*
a) Compensator top assembly
b) Tube holder subassembly
c) Housing subassembly
d) Spring holder subassembly

The main assemblies are simulated under maximum load, in which case the two subassemblies are also in physical contact with each other. This can occur at the specified maximum load capacity or during the exceeding of the maximum load capacity (overload) *(Figure 4)*. Four arrangements were selected from the eight types of the manufacture's products. The concerning manufacturer ID's are A, B, E, G.

At the maximum load or overload, the compensator is geometrically terminated to protect the spring inside [4].

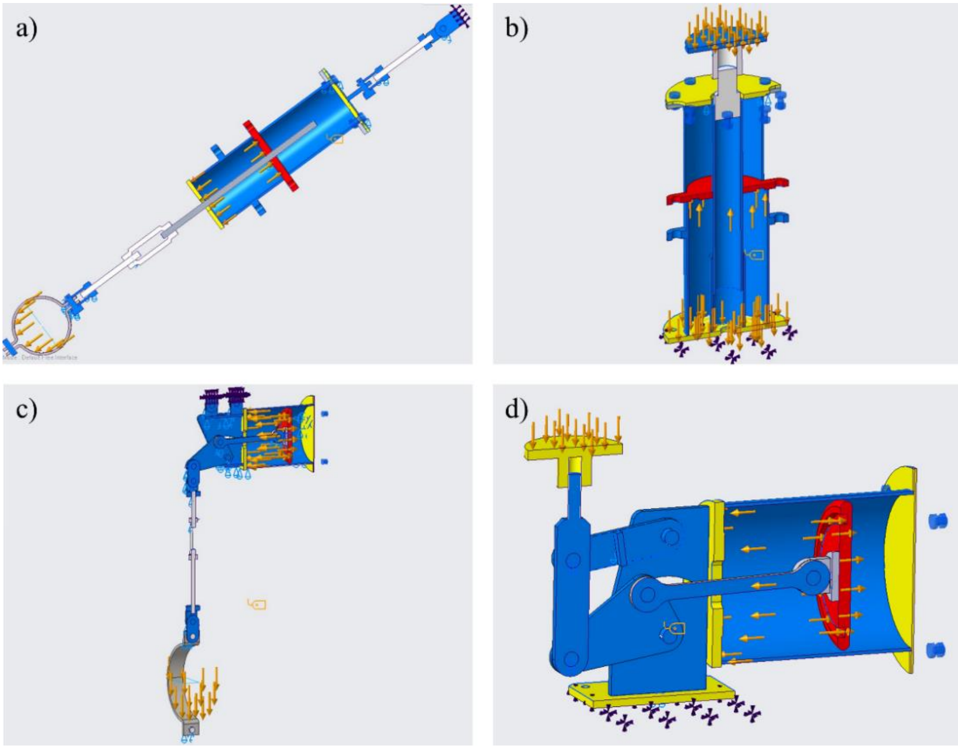

Figure 4. *Load cases of four different top assemblies*
a) Type A
b) Type B
c) Type E
d) Type G

Simulation of subassemblies in 4 positions:

- minimum working load (the spring plate just touches the spring housing cover but does not yet apply a load to it),
- intermediate load (half the range of movement of the spring plate),
- maximum working load (either the specified maximum load - the two subassemblies does not contact with each other; or the value below the specified maximum load – the two subassemblies just contacts with each other, there is no load transfer at the point of impact),
- overload (either the specified maximum load is exceeded – the two subassemblies contact with the help of an overload; or the specified maximum load – the two subassemblies are contacted with each other, there is a load transfer at the point of collision).

The static component was fixed (all degrees of freedom were 0) in each load case. The fasteners were substituted with special boundary conditions.

Illustration of the simulation result for main assemblies and subassemblies at overload *(Figure 5)*.

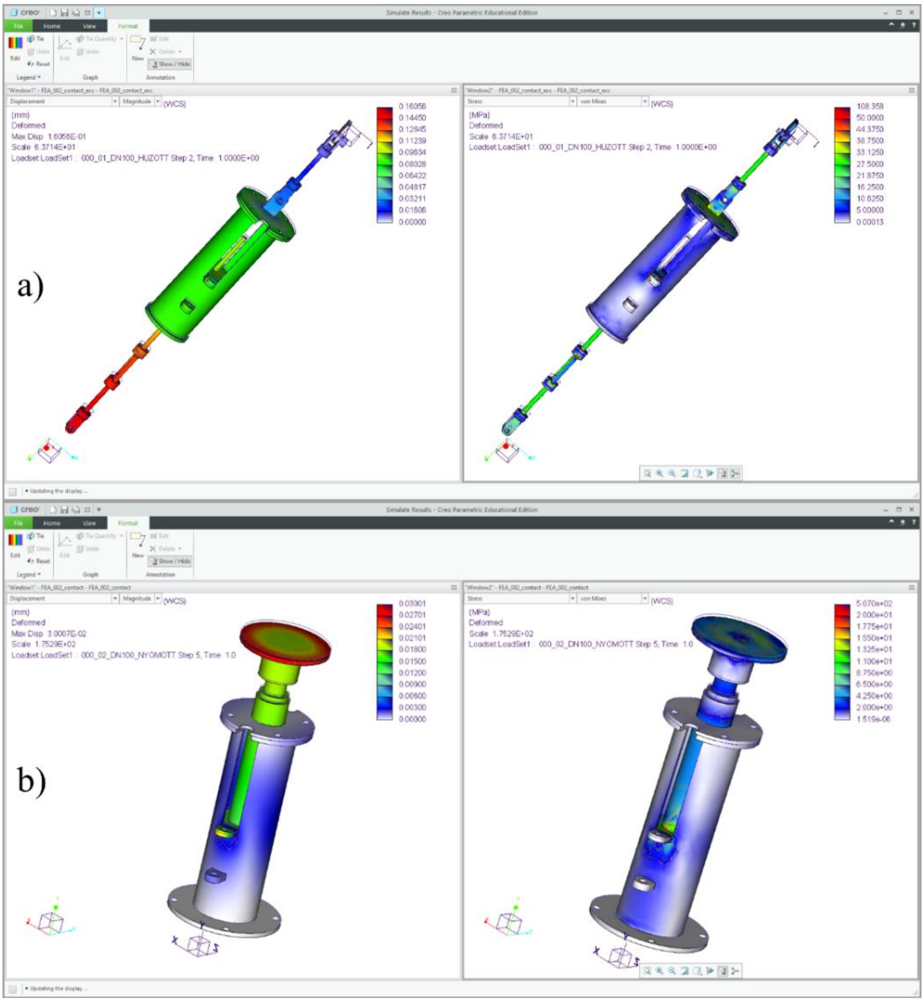

Figure 5. *FEA results in case of overload*
a) Type A
b) Type B

Simplified CAD models [5] were also created from the nine tube compensators, which designed as a reference point for the analytical calculation. The subassemblies were modelled and simulated as components *(Figure 6)*.

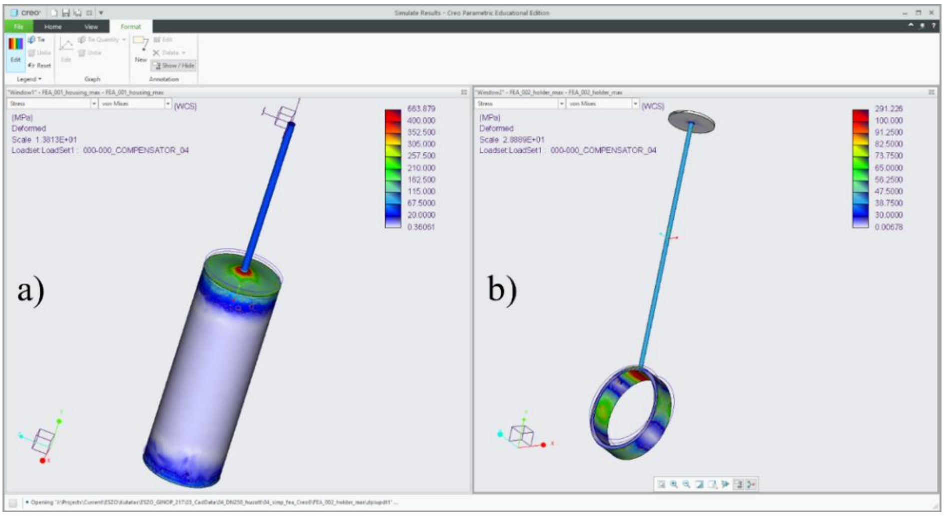

Figure 6. *FEA results on simplified CAD models*
a) Simplified model of housing
b) Simplified model of tube holder

Unfortunately, this method was not accurate enough. Therefore, the full simulations were used as a reference for the analytic calculations.

In parallel with the processes described so far, an Excel spreadsheet was continuously developed, in which, first, the data of the tube compensators were collected, and then various calculations were performed using these data.
In the four situations examined during the simulations, the following was determined:

- the amount of compression of the springs measured at rest (ΔL),
- the force required for the given compression (F_r),
- the length of the compressed spring measured under the given load (L).

The original and simplified models were also prepared. The deformation of the simplified models was calculated analytically as well, considering the main geometries (rod, tube) as spring models.

3. Results

The *Table 1* shows the results in summary, where the loads were given (F_{max}), and Max. Fr is the force required for the collision. For example, in the first case, it can be seen that the spring plate will contact the housing sooner than the maximum load. In the fourth case, it appears that it will not contact under maximum load, but only in the event of overload.

Table 1
Values of maximum loads

LOADS – until contact										
	min. ΔL	**min. F_r**	**rug. L**	**mid. ΔL**	**mid. F_r**	**rug. L**	**max. ΔL**	**max. F_r**	**rug. L**	**F_{max}**
1	65.0	545.0	350.0	157.5	1320.6	257.5	250.0	2096.2	165.0	3000
2	60.5	507.3	354.5	155.5	1304,0	259.5	250.5	2100.4	164.5	3000
3	23.0	26.5	257.0	116.0	133.6	164.0	209.0	240.8	71.0	1000
4	83.0	2093.7	460.0	174.5	4401.9	368.5	266.0	6710.0	277.0	6136
5	83.0	2093.7	460.0	174.5	4401.9	368.5	266.0	6710.0	277.0	6136
6	166.0	4447.8	357.0	214.3	5740.6	308.8	262.5	7033.4	260.5	7566
7	166.0	4447.8	357.0	214.3	5740.6	308.8	262.5	7033.4	260.5	7566
8	109.0	6449.8	444.0	198.5	11745.7	354.5	288.0	17041.6	265.0	14000
9	108.0	6390.6	445.0	175.5	10384.7	377.5	243.0	14378.9	310.0	14000
	[mm]	[N]	[mm]	[mm]	[N]	[mm]	[mm]	[N]	[mm]	[N]

These results are also clearly visible in the figures *(Figure 7)*. Where there is a fracture, the contact occurs because the system's rigidity changes significantly. This is spectacular in Example 3 since where the housing and the plate contact, there is a more significant fracture, and then the whole system stretches under the influence of force (F_{max}).

The research revealed that if we perform the simulation with significantly simplified geometry, the behaviour of the geometry will be very different so we will get very different results. However, calculations and simulations from the original model produce nearly identical results for elongation. This may be because the metal plate's elongation is not calculated, but it is included in the simulation.

Figure 7. *Comparison of simulated and calculated results*

The simplified analytical calculation is where geometries have been replaced by elongating rods or tubes. The housing was a tube, and the holder parts were rods. For example, *Figure 8* shows that only the holder will stretch in the calculations. In conclusion, the more straightforward analytic calculation was more proportional to the full detailed finite element simulation than the behaviour of the simplified finite element model.

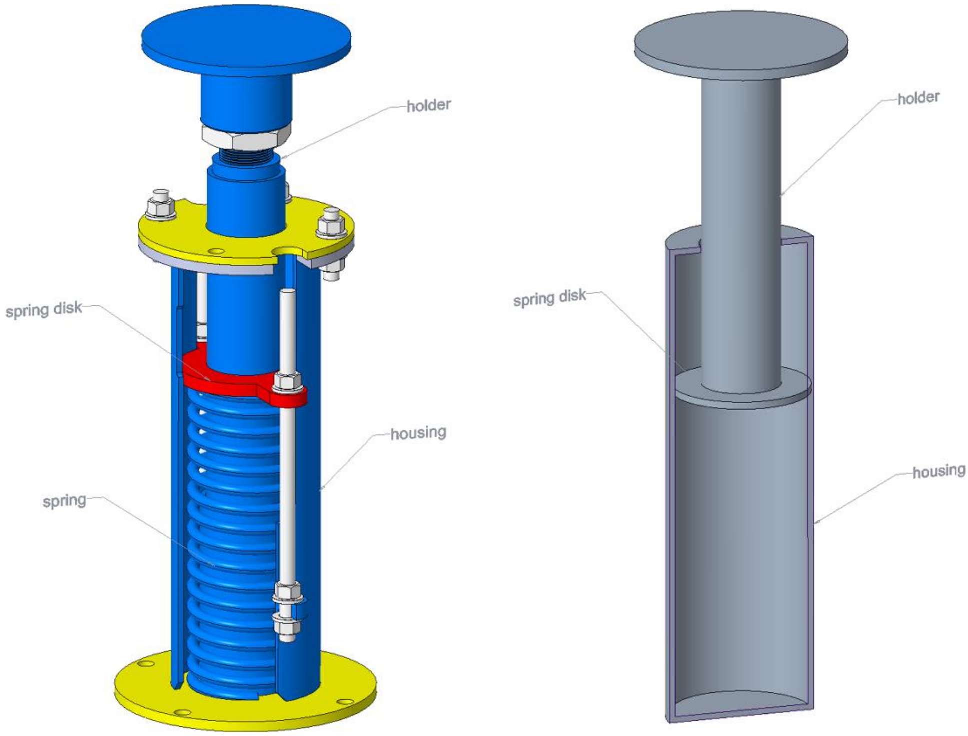

Figure 8. *Simplified geometry for analytic calculation*

Multiplying the calculated result by a specific factor gives the original values, which are 1.22 on average for pulling and 0.98 for pressure. When pulling, the deviation may be due to varying cross-sections. This result means that under a reasonable accuracy there is not needed to simulate the whole model, but the usage of analytical calculations provides accurate results for further design.

Based on these results, a design guide was also developed in SMath opensource software. The program used to perform the analytical calculations was of great help in calculating the desired mathematical operations using the appropriate formulas, instead of paper-based manual calculations. Arranging the four states detailed above into matrices could be performed as a single operation. Compared to paper-based calculations, the chances of human omission (accounting) could be significantly reduced. Not least, the formula, once made, could be applied to the nine types of compensators (with the help of copying), which significantly speeded up the calculations. SMath can also export a .exe application from the calculations, which allows you to change the parameters of a particular compensator type. The application performs the operations with the changed parameters and prints the result in the appropriate location.

4. Conclusion

In order to investigate the deformity of the tube holders, three steps were presented in this project: tube holder strength simulation in full detail, simplification (geometric transformations), simplified analytical calculations.

The result shows that there is no need to simulate the entire model, the application of analytical calculations provides accurate results for further design. In specific circumstances this simplified mathematical model substitutes the more complex FEA analysis well. This kind of calculation can be the calculation of displacements in a wide range tube network.

Acknowledgement

ESZO Kft.'s application for support under the Economic Development and Innovation Operative Program of GINOP registered with the identification number 2.1.7-15-2016-00764 was considered worthy of approval. Within the framework of the project, the company developed and then manufactured the pipe holder product family with the EMARUG name.

The research reported in this paper is part of project no. BME-NVA-02, implemented with the support provided by the Ministry of Innovation and Technology of Hungary from the National Research, Development and Innovation Fund, financed under the TKP2021 funding scheme.

References

[1] Mills, N. (2007). *Polymer Foams Handbook*. Butterworth-Heinemann, https://doi.org/10.1016/B978-0-7506-8069-1.X5000-4.

[2] Klobucar, R. – Acko, B. (2016). Experimental Evaluation of Ball Bar Standard Thermal Properties by Simulating Real Shop Floor Conditions. *International Journal of Simulation Modelling*, Vol. 15, No. 3. https://doi.org/10.2507/IJSIMM15(3)10.356.

[3] Stoll, H. W. (1999). *Product Design Methods and Practices*. CRP Press, Boca Raton, https://doi.org/10.1201/9781482276800.

[4] Knauf, G. – Kulgemeyer, A. (2015). *Major Standards for Line Pipe Manufacturing and Testing*. Wiley Online Library. https://doi.org/10.1002/9781119019213.ch16.

[5] Horváth, I. et al. (2005). *Advanced Design Support*. Delft University of Technology, ISBN 7-5062-7444-2.