

Dorian Nedelcu

Tensile Test Numerical Simulation with Finite Element Method

The paper compares calculus made by Cosmos M Design Star software with classical tensile calculus, statistical values and with experimental tensile test for standard tension test specimen.

1. Tensile tests

The tensile test conditions are set through [1]. On the tension test specimen figure 1, is applied a progressive stretching force P on its longitudinal direction, which stretches the tension test specimen continuously until it breaks.

Normal stress σ in the transversal section of the test specimen is given, indifferent of deformation grade:

$$\sigma = \frac{P}{S_0} \quad (1)$$

where $S_0 = 78.5 \text{ mm}^2$ is the area of the transversal section for diameter $d=10 \text{ mm}$.

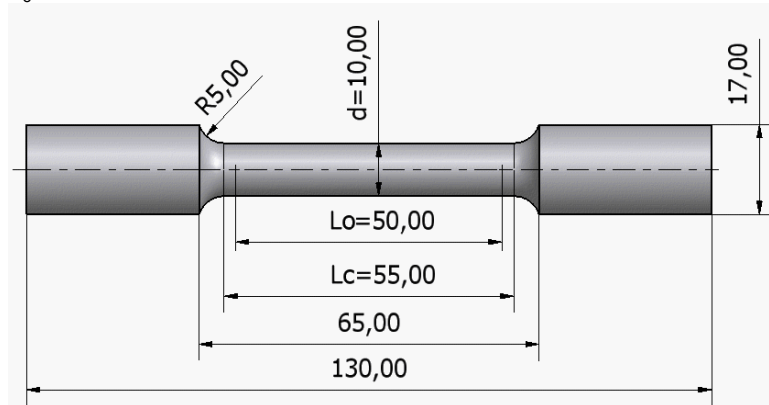


Figure 1. The tension test specimen dimensions subject of tensile
The deformation during the tensile test is characterised by strain ε and the area reduction ψ :

$$\varepsilon = \frac{L - L_0}{L_0} = \frac{\Delta L}{L_0} \quad (2)$$

$$\psi = \frac{S_0 - S}{S_0} \quad (3)$$

where, for the tension test specimen: L is the instant length between marks, $L_0 = 50$ mm is the initial length between marks; ΔL is the instant extension and S the instant area of the deformed test specimen.

During the tests, the values of the force P and the extension ΔL are followed by recording devices; the force speed must be constant and between the values $6 - 30$ N/mm²s. The tensile test diagram, figure 2, is made of the values pairs (ΔL și P). Usually, this diagram is automated printed by the device. Since, the S_0 and L_0 are constant values, the $\Delta L - P$ diagram is similar, on another scale, with tensile diagram $\sigma - \varepsilon$ of the material.

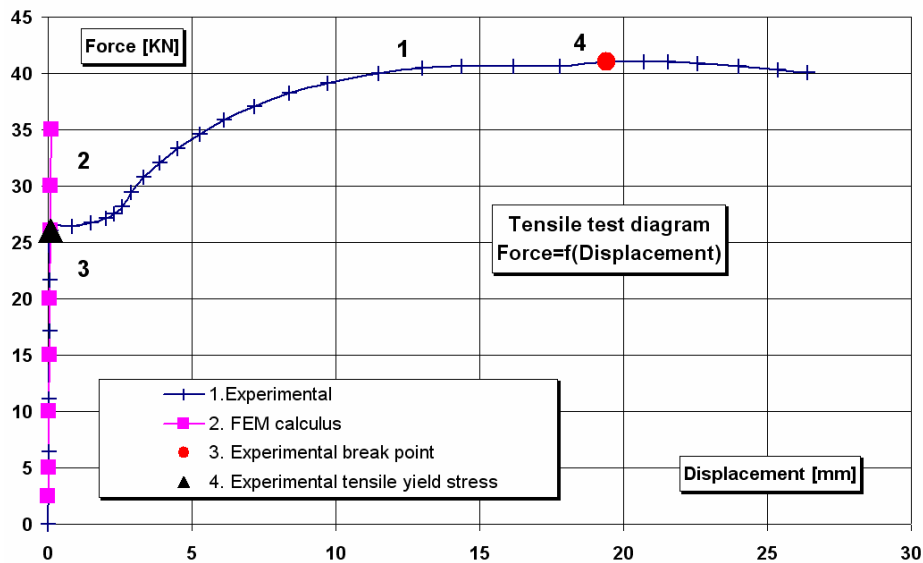


Figure 2. The tensile test diagram

For the tensile diagram, the following characteristics can be obtained:

- Tensile yield stress σ_c - defined as unitary stress where the test specimen length increase for the first time, without increasing the force P (point 3 figure 2):

$$\sigma_c = \frac{P_{cs}}{S_0} \quad (4)$$

- Tensile breaking strength σ_r - defined as maximal unitary stress for maximal force (point 4 figure 2):

$$\sigma_r = \frac{P_{\max}}{S_o} \quad (5)$$

- Breaking extension δ_n or A_n :

$$\delta_n = \frac{L_u - L_o}{L_o} 100 \quad [\%] \quad (6)$$

where L_u is the distance between marks, measured on united the broken parts of the test specimen.

- Breaking area reduction Z:

$$Z = \frac{S_o - S_u}{S_o} 100 \quad [\%] \quad (7)$$

where S_u is the area of minimal normal section of the test specimen measured after the test.

- Coefficient of elasticity E (Young's modulus) - defined as the rapport between the increasing of the unitary stress $\Delta\sigma$ and the increasing of the strain $\Delta\varepsilon$, in the elastic domain:

$$E = \frac{\Delta\sigma}{\Delta\varepsilon} \quad (8)$$

The tensile diagram was obtained for experimental tensile tests on the OL 52 steel, figure 1; table 1 present the comparison between experimental and statistical values [4].

Table 1

Regime	Yield		Break	
	Experimental values	Statistical values	Experimental values	Statistical values
The tensile force P [KN]	26	-	41	-
Tensile breaking strength σ_r [MPa]	331	360	522	520-560
The last distance between marks L_u [mm]	-		69.4	-
Breaking extension A_5 [%]	-	20	38.8	-
Minimal section after break S_u [mm ²]	-		74.9	-
Breaking area reduction Z [%]	-		5	-

2. Tensile Tests Numerical Simulation with Finite Element Method

The calculus was made with Cosmos M Design Star software [3], [5]. When loads are applied to a body, the body deforms and the effect of loads is transmitted throughout the body. The external loads induce internal forces and reactions to render the body into a state of equilibrium.

Linear Static analysis calculates displacements, strains, stresses, and reaction forces under the effect of applied loads.

Linear static analysis makes the following assumptions:

- *Static Assumption* - all loads are applied slowly and gradually until they reach their full magnitudes. After reaching their full magnitudes, loads remain constant (time-invariant).
- *Linearity Assumption* - The relationship between loads and induced responses is linear. For example, if you double the loads, the response of the model (displacements, strains, and stresses), will also double. You can make the linearity assumption if:
 - all materials in the model comply with Hooke's Law, that is Stress is directly proportional to Strain.
 - the induced displacements are small enough to ignore the change in stiffness caused by loading.
 - boundary conditions do not vary during the application of loads. Loads must be constant in magnitude, direction, and distribution. They should not change while the model is deforming.

The internal forces in a body vary from one point to the other. Across any small internal plane area, loads are exerted by the part of the body on one side of the area upon the part on the other side. Stress denotes the intensity of these internal forces (force per unit area).

Finite Element Analysis (FEM) provides a reliable numerical technique for analyzing engineering designs. The process starts with the creation of a geometric model. Then, the program subdivides the model into small pieces of simple shapes (elements) connected at common points (nodes). Finite element analysis programs look at the model as a network of discrete interconnected elements.

FEM assumes that the behavior of each element varies in particular known fashions for various conditions. The Finite Element Method (FEM) predicts the behavior of the model by manipulating the information obtained from all the elements making up the model.

When analyzing a part or assembly, we need to investigate its response to various scenarios of service environments and operational conditions.

A *design study* represents a simulation of a "what-if" scenario, which is completely defined by 1) the type of analysis and related options, 2) material assignments, 3) loads and boundary conditions, and 4) a mesh.

To study the impact of varying any of the elements defining a study, we can create a new study or modify an existing one. We can create a number of studies with different materials, loads, boundary conditions, and meshes.

The numerical simulation was made for 8 forces, with increasing values applied to the same 3D model: $P=2.5, 5, 10, 15, 20, 26, 30, 35$ KN.

For FEM calculus is considered only the cylindrical geometry of the test specimen with the diameter $d=10$ mm and $L_0=50$ mm (figure 1), to avoid affecting the simulation results by the geometry destined to hold the test specimen.

With **Open** option from **File** menu the 3D geometry, figure 1, generated with Autodesk Inventor [2], will be loaded into Cosmos M Design Star the file Epruveta FEM.ipt.

There was created 8 linear static analysis study, corresponding to 8 values for force P and with solid mesh. The material selected was Alloy Steel, with the following characteristics: coefficient of elasticity $E=2 \times 10^5$ N/mm² and Poisson's ratio $\nu = 0.29$.

The cylindrical part of the test specimen is fixed in the origin, with length in X direction, by imposing 0 values for X translation and force P will be applied on the opposite side, figure 3.

The geometry was meshed in 52850 finite elements with 75906 nodes and a medium size of finite elements equal to 0.78917 mm, figure 4.

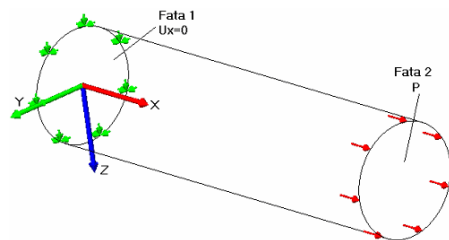


Figure 3 Loads applied to the geometry

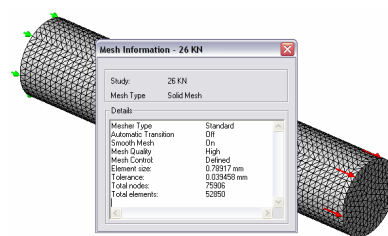


Figure 4 Mesh parameter

The geometry was meshed in 52850 finite elements with 75906 nodes and a medium size of finite elements equal to 0.78917 mm, figure 4.

After run, in **Visualizer** zone will be displayed the results **Stress, Displacement, Strain, Deformation** și **Design Check**, which can be used to visualize the numerical and graphical results of the study. The FEM results are numerically presented in table 2, for 8 test studies, simultaneously with the theoretical values for stress, calculated with formula 1 and also displacement calculated with formula:

$$\Delta L = \frac{P \cdot L}{E \cdot S_0} \quad (9)$$

Figure 5 ... 12 show the graphical results for Von Mises stress and displacements obtained for P=5, 15, 26 and 30 KN. The reserved paper space is not enough to show all the graphical results.

Table 2

The tensile force P	FEM Values			Theoretical values	
	σ_{VonMises}	Displacement	Procentual elongation	σ	Displacement
[KN]	[MPa]	[mm]	[%]	[Mpa]	[mm]
2.5	31.96	0.0080	0.0159	31.85	0.0080
5.0	63.93	0.0159	0.0318	63.69	0.0159
10.0	127.90	0.0318	0.0637	127.39	0.0318
15.0	191.80	0.0478	0.0955	191.08	0.0478
20.0	255.70	0.0637	0.1273	254.78	0.0637
26.0	332.40	0.0828	0.1655	331.21	0.0828
30.0	383.60	0.0955	0.1910	382.17	0.0955
35.0	447.50	0.1114	0.2228	445.86	0.1115

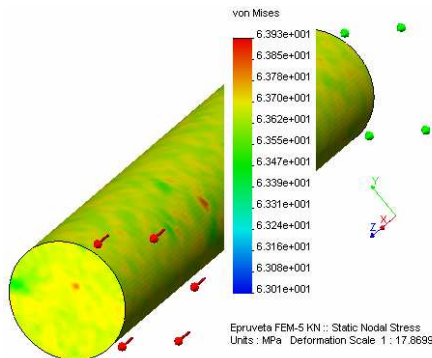


Figure 5 Von Mises Stress for P=5 KN

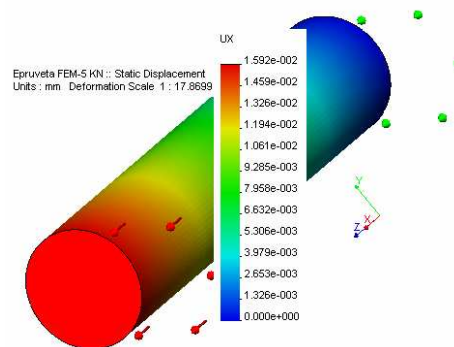


Figure 6 The displacements for P=5 KN

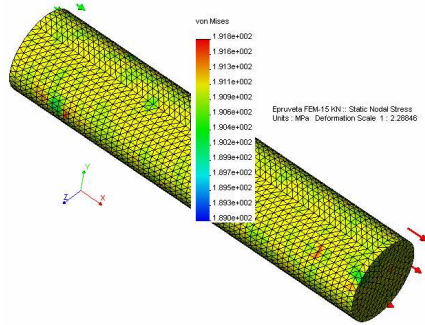


Figure 7 Von Mises Stress for P=15 KN

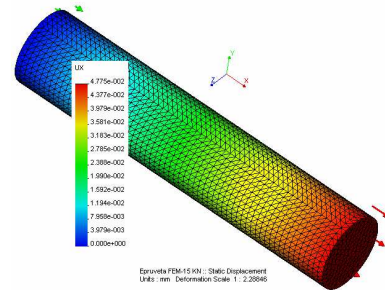


Figure 8 The displacements for P=15 KN

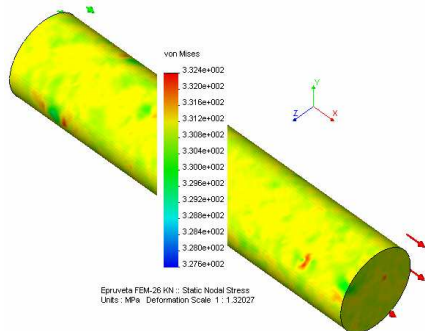


Figure 9 Von Mises Stress for P=26 KN

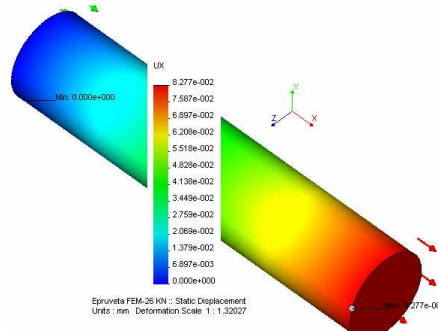


Figure 10 The displacements for P=26 KN

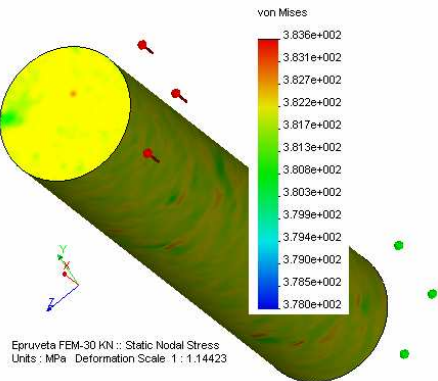


Figure 11 Von Mises Stress for P=30 KN

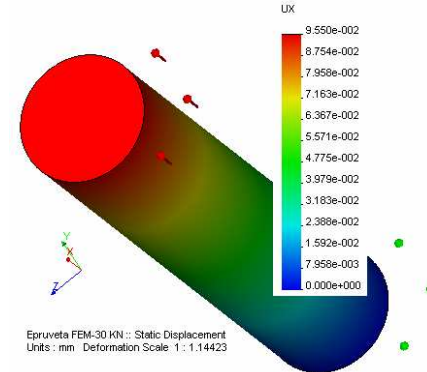


Figure 12 The displacements for P=30 KN

3. Conclusion

From numerical and experimental values results the following conclusions:

- the numerical values calculated with FEM are the same with theoretical values (table 1);
- the experimental curve force as function of displacement is presented in figure 1 (curve no. 1); the same curve for FEM calculus is presented in figure 1 (curve no. 2); can be observed that the two curves are identical, until the value of tensile yield stress figure 1 (point no. 3); after that point the values are completely different; the reason of this difference is the consequence of the linearity assumption (Hooke's law) which is admitted in FEM calculus beyond the tensile yield stress value, which is not true, because after that point the Hooke's law is not applied;
- until the value of tensile yield stress the numerical and experimental results are the same for tensile test; this conclusion justify the FEM calculus, before the tensile yield stress value; passing over that value impose a nonlinear analysis.

References

- [1] ***** *Materiale metalice. Încercarea la tracțiune*. SR En 10002-1.
- [2] Nedelcu D. *Modelare parametrică prin Autodesk Inventor*, Editura „Orizonturi Universitare” Timișoara, ISBN 973-638-116-1, Mai, 2004
- [3] Tiberiu Ștefan Mănescu T.S., Nedelcu D. *Analiză structurală prin metoda elementului finit*, Editura „Orizonturi Universitare” Timișoara, ISBN 973-638-217-6, Octombrie, 2005.
- [4] Buzdugan G. *Manualul inginerului mecanic*, Editura Tehnică, București, 1973
- [5] ***** *Design Star User Guide Reference Manuals*

Address:

Assoc. Prof. Dr. Eng. Dorian Nedelcu, “Eftimie Murgu” University of Reșița,
Piața Traian Vuia, nr. 1-4, 320085, Reșița, d.nedelcu@uem.ro