11th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2007, Hammamet, Tunisia, 05-09 September, 2007.

STRESS ANALYSIS OF MODEL DEVICE FOR TORSION

A. Muminović; I. Šarić; N. Repčić
Faculty of Mechanical Engineering Sarajevo
Vilsonovo šetalište 9, 71000 Sarajevo
Bosnia and Herzegovina

ABSTRACT
On the example of stress analysis of model device for torsion, during the work, justification of Finite Element Method (FEM) application was shown by use of CAD/CAM/CAE software system CATIA V5 in the case when it is impossible to make a prototype in order to do its experimental testing. First, stress analysis of real model device was done by experimental tensometric method, and after that stress analysis was done by method of device modelled in CATIA V5 system. HBM digital measuring amplifier DMC 9012A were used for stress analysis done by experimental tensometric method, while stress analysis done by FEM method was done in Generative Structural Analysis module of CATIA V5 system. Justification of FEM method application for stress analysis, in the case when it is impossible to make a prototype, is shown by the same results of done stress analyse.
Key words: stress analysis, FEM, CAD/CAM/CAE, CATIA V5

1. INTRODUCTION
Two methods were done for stress analysis of model device for torsion: experimental tensometric and FEM method, and after that the results of analysis were compared (table 2., figure 6.). Analysis was done for three different cases of lever load, length $l = 300$ mm, by force ($F = 9{,}81$ N, $F = 49{,}05$ N and $F = 98{,}1$ N) in negative direction of z axis.

2. STRESS ANALYSIS DONE BY EXPERIMENTAL TENZOMETRIC METHOD
Measuring of stress values by experimental tensometric method, at the measuring spot of axle which is 60 mm away from support (figure 1.b), was done by HBM digital measuring amplifier DMC 9012A (figure 1.a). Measuring process consists of changing of construction characteristic parts dilatation into measurable electric value, which is achieved with measuring tapes.

a) b)

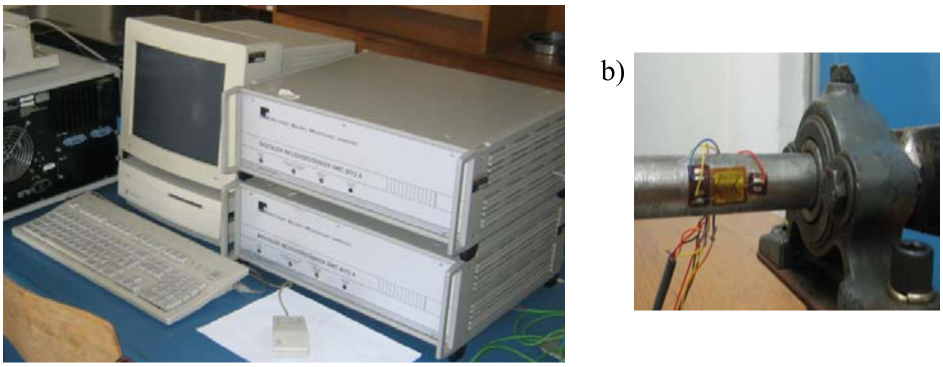

Figure 1. HBM digital measuring amplifier DMC 9012A and analised measuring spot of modele device for torsion

Measuring tapes are placed under the angle from 45° from the lever axis where maximal value of tangential stress is expected. Principled scheme of the measuring tapes position and dilations measuring at measuring spot is shown in the figure 2.

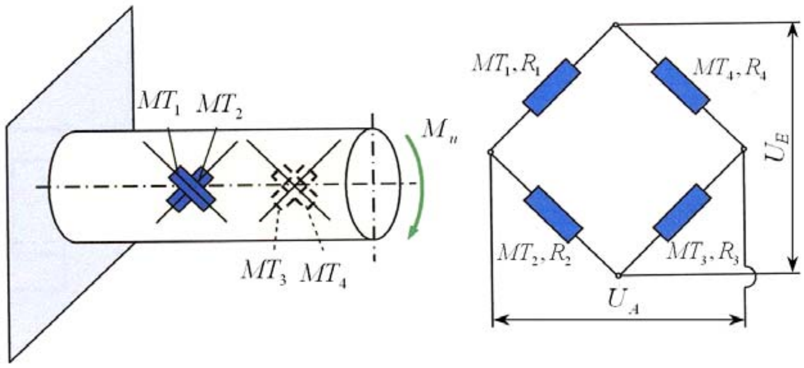

Figure 2. Principled scheme for stress defining with measuring tapes

As the measuring tapes principle means electric values changing, load change provoked stress change in time, and diagrams for given loads are shown in the figure 3.

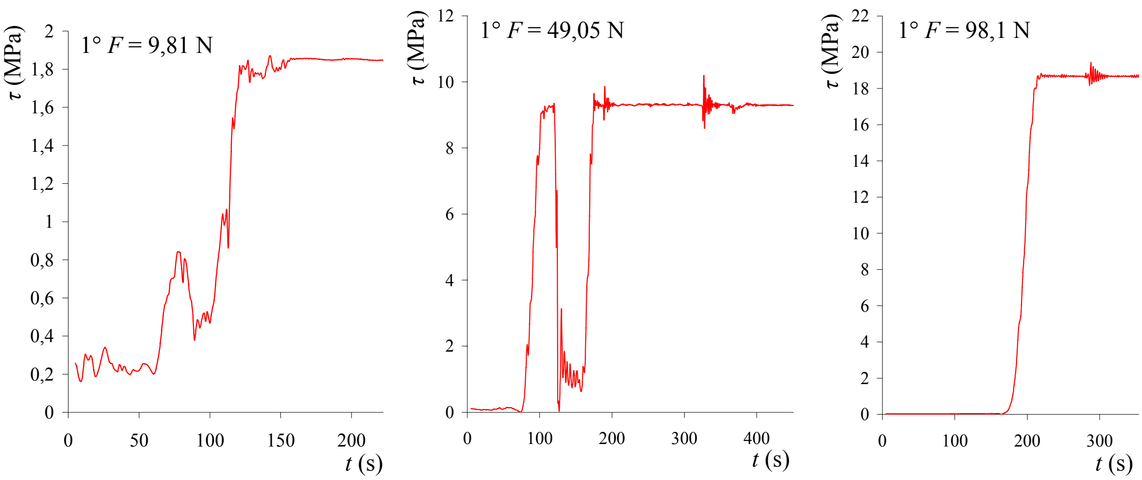

Figure 3. Diagrams of stress changing in time

Transforming of obtained values mV/V into MPa is done on the following equation base:

$$\varepsilon_{45^\circ} = \frac{U_A}{U_E} / K_t, \qquad \text{... (1)}$$

where is: ε_{45° - longitudinal extension under the angle $\pm 45^\circ$ from direction of load, $\frac{U_A}{U_E}$ - obtained measuring values and $K_t = 2$ - coefficient of tape for used type.

Value of stress is set on the formula base:

$$\tau = 2 \cdot G \cdot \varepsilon_{45^\circ} \text{ (MPa)}, \qquad \text{... (2)}$$

where is: G (MPa) - modulus of elasticity in shear.

3. STRESS ANALYSIS DONE BY FEM METHOD

In majority of cases, Finite Element Method (FEM) is used for prototypes for which results of experimental testing have already existed, in order to check FEM results and, if it is necessary, to do some changes of construction. Analysis done by FEM method is a technique, supported by computer, for stress and deformations fixing within tested structure. Structure (geometry of part) is approximated by finite number of mutual connected standardized shapes elements (triangles, quadrangles, volume elements and similar) for which mathematics solution can be found, and after that global problem is being solved by linking method. Structure prepared in this way is being load with exterior forces or temperatures (pre processing). Stress analysis of modelled device done by FEM method was done in *Generative Structural Analysis* module of CATIA V5 system. The first step is the modelling device in *Sketcher*, *Part Design* and *Assembly Design* modules of CATIA V5 system. Later, model divides into finite number of elements and represents in common mesh coordinate system. Elements are linked in nodes through which output data are obtained. Geometry of parts approximates with mesh elements, but areas of parts and mesh are not completely identical.

Table 1. Mesh characteristics, type of elements and maximal load resultants

Mesh		Element type		Maximal load resultants	
Entity	Size	Connectivity	Statistic		
Nodes	60888	TE10	21667 (16,88%)	F_x = 1,4901e-008 N	M_x = 2,7043e+001 Nm
Elements	128325	TE4	99801 (77,77%)	F_y = 2,9802e-008 N	M_y = -1,5112e+001 Nm
		SPIDER	6857 (5,34%)	F_z = -9,8100e+001 N	M_z = -6,2092e-010 Nm

For stress analysis doing by FEM method, for modeled device, it needs to define: material of all parts, leaning, load and links between the parts in assembly block. After that, estimate, review (post processing) and analysis of results are done. In the figure 4. Von Mises stress values on some structure parts for the case of maximal load with force $F = 98{,}1$ N are shown.

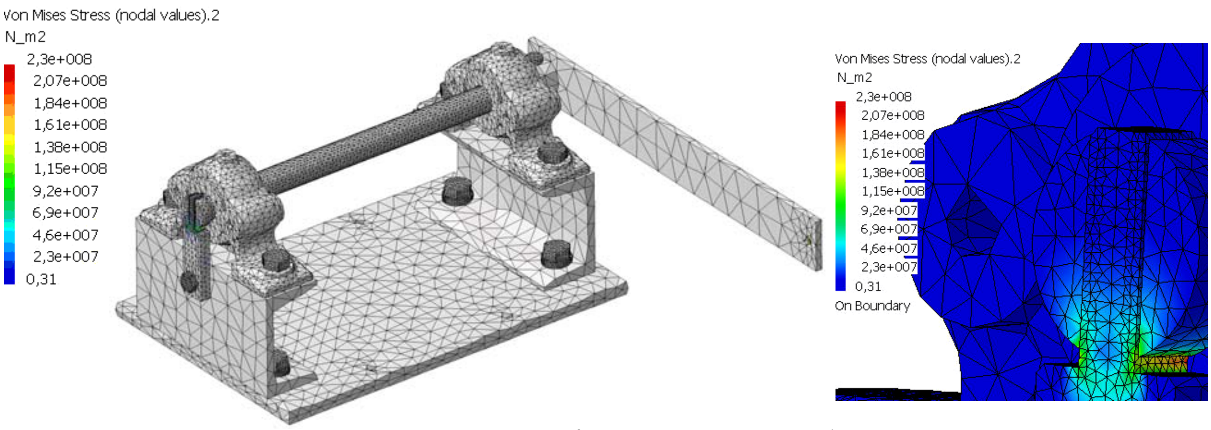

Figure 4. Review of Von Mises stress and spot with maximal stress values

In order to compare with results obtained by experimental tensometric method use, measuring spot was analysed, too, for three different cases of load by force (figure 5.).

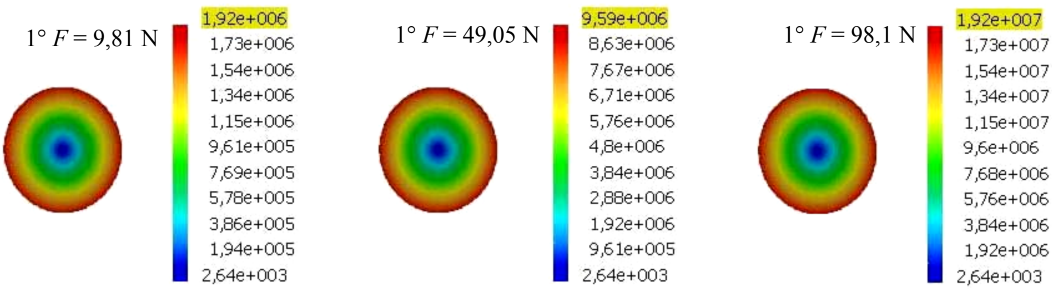

Figure 5. Tangential stress values in cross section of axle

4. COMPARATION OF STRESS ANALYSE RESULTS

Table 2. and figure 6. confirm identical stress analysis results of experimental tensometric and FEM method.

Table 2. Results of done stress analysis

Load F (N)	Tangential stress values τ (MPa) – Method: Experimental tensometric	Tangential stress values τ (MPa) – Method: FEM
9,81	1,85	1,92
49,05	9,23	9,59
98,1	18,66	19,2

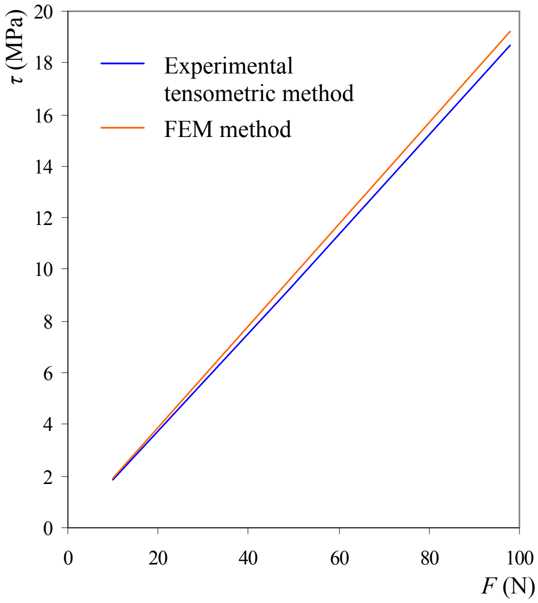

Figure 6. Results comparing of done stress analysis

5. CONCLUSION

Stress analysis of mechanical constructions the most frequently means experimental prototype testing. However, prototype making is in most cases very complicated and expensive, and because of that, in recent years, a large number of close specialised software systems for quality carrying out of mechanical constructions stress analysis have been used. One of such systems is software system CATIA V5, which module *Generative Structural Analysis* was used during FEM analysis of modelled device for torsion. In spite of the fact that CAD/CAM/CAE software systems require skill staff and computer equipment of certificated configuration, results that they give consider in the top of stress analysis in order to reduce experimental testing in the early phase of product development. These and many other reasons lead to the fact that software systems, which treat stress analysis of mechanical constructions, acquire a great popularity. Graphical possibilities and ability for quick and simple data processing allow designer to finish and test their ideas interactive in real time, without any need for real prototypes making like during conventional experimental testing.

6. REFERENCES

[1] Zienkiewicz O.C., Taylor R.L., Zhu J.Z.: The Finite Element Method: Its Basis and Fundamentals, 6th edition, Butterworth-Heinemann, Oxford, 2005.,

[2] Amirouche, F.: Principles of Computer-Aided Design and Manufacturing, 2nd edition, Prentice Hall, Upper Saddle River, New Jersey, 2004.,

[3] Spotts M.F., Shoup T.E., Hornberger L.E.: Design of Machine Elements, 8th edition, Prentice Hall, Upper Saddle River, New Jersey, 2004.,

[4] Shigley J.E., Mischke C.R., Budynas R.G.: Mechanical Engineering Design, 7th edition, McGraw-Hill, New York, 2004.,

[5] Zienkiewicz O.C., Taylor R.L.: The Finite Element Method – Volume 2: Solid Mechanics, 5th edition, Butterworth-Heinemann, Oxford, 2000.,

[6] Repčić, N.: CAD Methods: Computer-Aided Design, Faculty of Mechanical Engineering, Sarajevo, 1996.