20th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2016, Mediterranean Sea Cruising, 24th September - 1st October, 2016

RELIABILITY OF NUMERICAL MODELING AND SIMULATION OF TEMPERED GLASS PLATE FOR LOAD

Mileta Janjić
Univesity of Montenegro - Faculty of Mechanical Engineering
Cetinjska 2, Podgorica
Montenegro

Dragana Đurašković
"DM KRISTAL" LLC
Vasa Raičkovića 29, Podgorica
Montenegro

ABSTRACT
Application of numerical modeling and simulation has proved very useful and has become an irreplaceable tool in solving various technical problems, at all stages of research, development, design, construction, production, operation and maintenance of products and systems of different physical nature. The aim of this paper is to show the reliability of the numerical modeling and simulation in load of tempered glass plate by the finite element method (FEM). At the same time the software package ANSYS is used. At the end is made a comparative analysis of numerical and experimental results, which proved to be justified, the reliability of the numerical simulations performed on the computer.
Keywords: Finite element method, ANSYS, numerical modeling, simulation, stress, strain.

1. INTRODUCTION
The use of computers in the engineering practice caused the transition from analytic into numerical methods of calculations. The finite element method has shown the most utilization value of all the numerical methods. The FEM is the most known and most used method for numerical solutions of observed real problems [1,2].
The contemporary architecture recognizes glass as a practically indispensable material, both in building of interior and exterior structures. Glass found its use at places where other materials were used earlier. Tempered glass is up to five times stronger than simple glass what recommends it for manufacturing of glass doors, fences, eaves, i.e. wherever there is a danger of possible injuries caused by fractures. The goal of this research work is dedicated to studying of the stress-strain parameters of tempered glass panels and the reliability of the numerical simulation method by use of the finite element method. The software suite ANSYS will be used for this. The comparative analysis of numerical results and results obtained by the experimental researches will be finally conducted [3].

2. MODELLING AND SIMULATION OF STRESS OF TEMPERED GLASS PANEL
The topic of modelling by the finite element method in the ANSYS software suite is a tempered glass panel with thickness of 4 [mm]. The characteristic stress distribution emerge in tempered glass: molecules on the surface are permanently exposed to stress, while internal molecules are exposed to tensile stresses. These stresses must be balanced because this is a condition for a stabile state ensuring the adequate safety properties of tempered glass [3,4]. The characteristics of glass: density, Jung's elasticity module and Poisson's number were taken from the reference sources and were applied as the entry parameters for the simulation.

Table 1. Entry parameters for ANSYS simulation of tempered glass 4[mm]

Geometry	Graphics Properties	Visible	Yes
		Transparency	1
	Definition	Suppressed	No
		Material	Glass
	Bounding box	Length x	500 [mm]
		Length y	4 [mm]
		Length z	150 [mm]
	Properties	volume	3.e+005[mm^3]
		mass	0.75 [kg]
Mesh	Statistics	Nodes	21653
		elements	3000
	Defaults	Global control	basic
		relevance	0
Contact	Generate contact on update		Yes
	Tolerance type		slider
	Tolerance slider		0
	Tolerance value		1.315 [mm]
	Face / face		Yes
	Face /edge		No

Environment	Definition	Ref. temp.	22°C
		inertia	off
	Weak Springs Reaction Force	Total	4.2692e-002 [N]
		X components	-3.2612e-005 [N]
		Y components	4.2692e-002 [N]
		Z components	-2.5739e-007 [N]
		X comp.	0 [N]
		Y comp.	-550 [N]
		Z comp.	0 [N]
Material	Glass	Density [Kg/m^3]	2500
		Jung's elasticity module [MPa]	82000
		Poisson's Number	0.23
	Steel	Density[kg/m^3]	7850
		Jung's elasticity module [MPa]	200 000
		Poisson's number	0.3

3. RESULTS OF FEM SIMULATION IN SOFTWARE SUITE ANSYS

Based on the results of set entry data by a numerical simulation, using the FEM in ANSYS, the results for stress, deformations and bending of a tempered glass panel with a thickness of 4[mm] were obtained. The exit results obtained by the modelling and simulation in the ANSYS, besides the tabular form, are also presented by help of the three-dimensional diagrams in colors of all parameters obtained by the modelling and simulation by the finite element method (Figure 1. - Figure 4.).

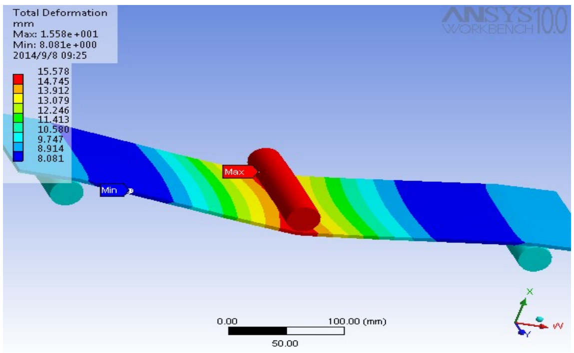

Figure 1. Deflection of tempered glass panel of 4 [mm]

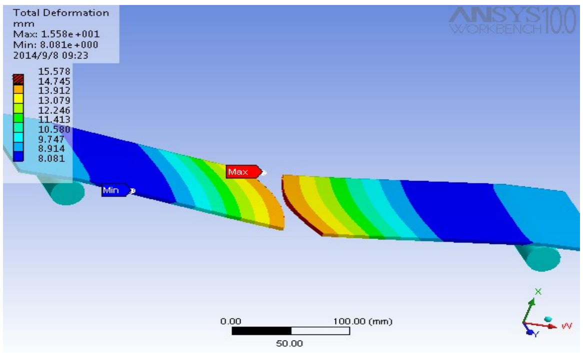

Figure 2. Fracture point of tempered glass panel of 4[mm]

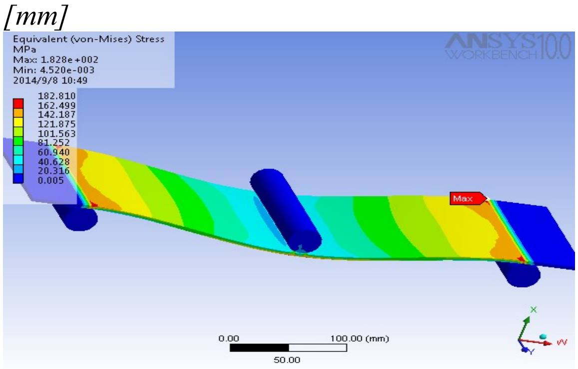

Figure 3. Distribution of von-Mises's stresses for tempered glass panel of 4 [mm]

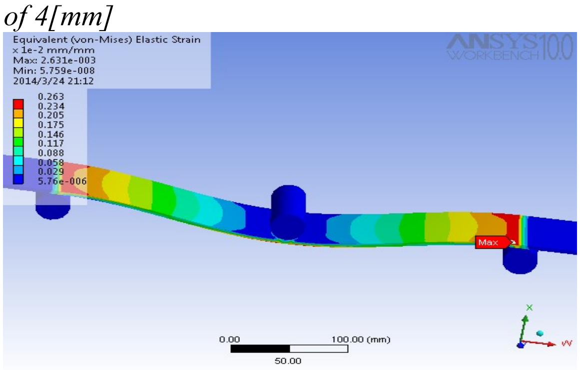

Figure 4. Distribution of von-Mises's deformations for tempered glass panel of 4 [mm]

The obtained results were elaborated in a further creation of the work by the drawing of charts of reliance between some weights, as well as the diagrams where the values of bending, stresses and deformations for tempered glass, as the subject of this paper, were compared.

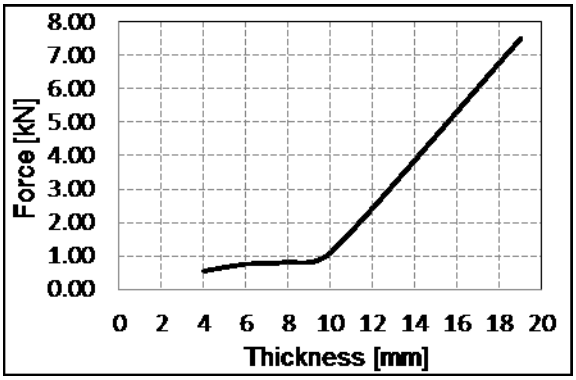

Figure 5. Force in function of thickness for tempered glass

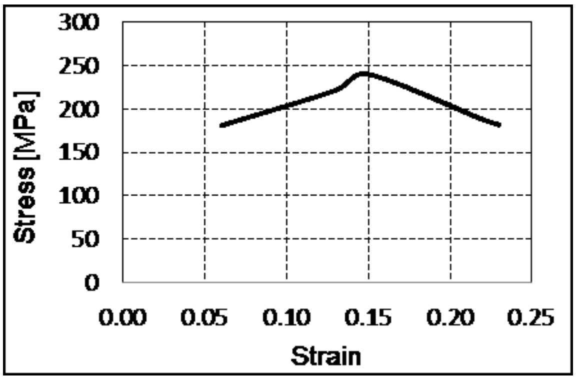

Figure 6. Diagram of dependence of stress and deformations for tempered glass

Figure 5 presents the load by a force in the function of a thickness for tempered glass. The diagram shows that, for a crack of glass panel thick up to 10 [mm], the rise of force is negligible, while for thicker panels this value immediately rises, as shown by the immediate curve jump. The diagram (Figure 6), shows the rise of stress for smaller deformations, and then a decline due to the rise of deformations.

4. COMPARATIVE ANALYSIS OF NUMERICAL AND RESULTS OBTAINED BY EXPERIMENTAL RESEARCHES

Simulations are very convenient tool, both from the aspects of scientific researches and for the solving of certain concrete problems of engineering practice. However, the question of their reliability always arises. Just because of it, the comparison of the value of loaded tempered glass panel, obtained by the modelling and simulation in software suite ANSYS with the experimental results [3]. The analysis of results is carried out in order to obtain the adequate data and to compare them. From Figure 7, seen almost entirely overlapping of stresses for thicknesses 4, 6, and 8 [mm] obtained experimentally and in ANSYS simulation can be seen, while for the thickness of 10 [mm], a fairly bigger stress, obtained by ANSYS simulation, was noticed.

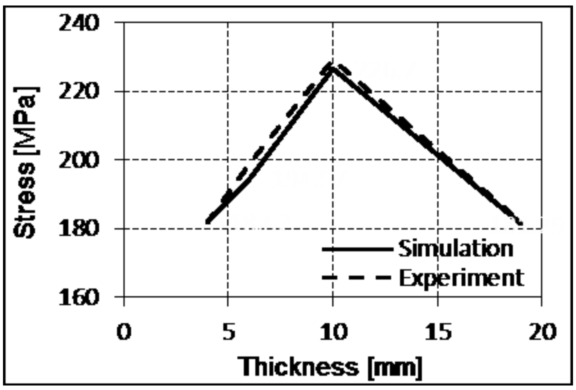

Figure 7. Stresses obtained by ANSYS simulation and experimental testing for tempered glass

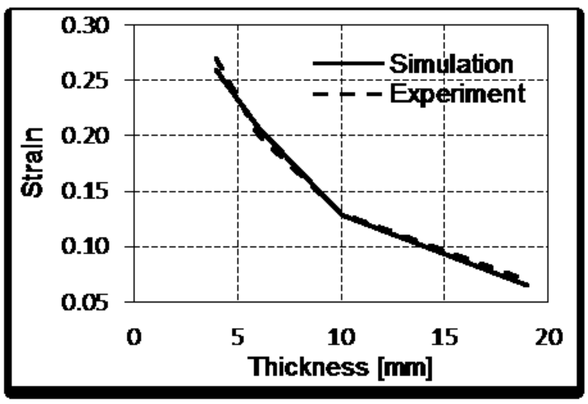

Figure 8. Strains obtained by ANSYS simulation and experimental testing for tempered glass

Figure 8. gives the comparison of results for deformations for tempered glass of 4, 6, 10 and 19 [mm]. The biggest deflections emerged for tempered glass of 4 [mm], while for other thicknesses a deflection almost is not observed. Therefore, the results of numerical simulation well overlap with real values when considering the simulation of deflection, stress and deformations, that could be seen from the graphic comparison of results. Based on the mentioned example, it can be stated that the

modelling and simulation of a stress – deformational state of a loaded tempered glass panel can be successfully carried out with the software suite ANSYS

5. CONCLUSION

The ANSYS software suite was used for obtaining of the results of numerical simulation by the use of the finite element method (FEM) for a loaded tempered glass panel, and then its reliability related to the solving of certain problems such as the emergence of bending, stresses and deformations was tested. After the comparison of these results with the results of the experimental research, a negligible bias of the values obtained by the modelling of the mentioned panel in software suite ANSYS by the finite element method was noticed. In a simulation of stress of a tempered glass panel in the ANSYS suite by the finite element method, the panel is being bent while it breaks. The size of bending depends on glass thickness, as can be seen on the diagram at Figure 5. For thinner glasses up to 10 [mm], it is necessary to act by a fairly smaller force, while a force causing breaking proportionally rises with rise of thickness. The difference between the values of bending from a simulation and values obtained by laboratory researches is almost negligible, thence the obtained results can be considered as satisfactory. Besides a bending, the numerical method in ANSYS gave also the values of stresses and deformations of a loaded glass panel. The diagram clearly shows a negligibly smaller deflection of the values obtained for stress than for deformations. The deflections emerged by modelling and simulation in the finite element method, by help of a computer, of all values of the research are negligible what ascertained the reliability of this method. This justified bigger and bigger use of computers in all research phases, without a loss of time and means for the creation of functional models, prototypes and their testing, bearing in mind that the use of narrowly specialized packets for problems that are topic of research must be taken into account.

6. REFERENCES

[1] Brčić S.: Metod konačnih elemenata-Departman za Tehničke nauke, Novi Pazar, 2014/2015.

[2] Costa S., Miranda M., Varum H., Teixeira-Dias F.: On the Evaluation of the Mechanical Behaviour of Structural Glass Elements, University of Aveiro, Portugal, 2006.

[3] Đurašković D.: Simulacija naprezanja i istraživanje profitabilnosti i održavanja pri obradi staklenih ploča - Magistarski rad, Mašinski fakultet, Podgorica, 2014.

[4] Hajdinak R.: Gradimo staklom, Reflex d.o.o, Podgora, 2009.

[5] Janjić M.: Istraživanje naponsko deformacionih parametara u procesima zapreminskog deformisanja - Doktorska disertacija, Mašinski fakultet, Podgorica, 2005.

[6] Janjić M., Đurašković D.: Modeliranje i simulacija naprezanja kaljenih staklenih ploča metodom konačnih elemenata, XXXIX Jupiter konferencija, Beograd, 2014.

[7] Jovanović J.: "Konstruisanje podržano računarom", Mašinski fakultet, Podgorica, 2013.

[8] Mandić V.: Fizičko i numeričko modeliranje procesa obrade deformisanjem, Kragujevac, 2012.

[9] Marjanović Z., Brzaković R.: Modeliranjem i računarskom simulacijom do boljeg kvaliteta proizvoda, Institut za automobile ZA, Kragujevac, 2006.

[10] Nedić N., Pršić D.: Računarsko modeliranje i simulacija u savremenoj tehnologiji upravljačkih komponenti i sistema, Mašinski fakultet Kraljevo i Mašinski fakultet Srpsko Sarajevo, Infoteh, Jahorina, 2001.

[11] Vrekić S., Živković M., Milovanović M.: Primjena metode konačnih elemenata u proračunima tankozidnih nosač, Mašinski fakultet, Kragujevac, Juinfo, Kopaonik, 2006.