FEM, numerical analysis, crack propagation, Abaqus, ABS

Patryk RÓŻYŁO[*], *Łukasz WÓJCIK*[**]

COMPARISON OF NUMERICAL AND EXPERIMENTAL ANALYSIS OF THE CRACK PROPAGATION PROCESS

Abstract

Nowadays numerical methods are a powerful tool to simulate real processes. Polymer materials are increasingly used in engineering solutions. ABS is a common material used interchangeably with respect to traditional materials. Process for the production of components for research on 3D printers is becoming more widespread and accessible to ordinary users. Crack propagation process was carried out at the same time to compare results through experiments performed on the testing machine and numerical study. Cracking process was carried out by finite element method, implemented on the basis of simulations xFEM. This method allows to conduct research related to the separation of the material nodes, in places of highest stress levels that exceed the limits of plasticity and strength of the material. The analysis involved comparison of the shape of the resulting cracks through experimental and numerical way.

1. INTRODUCTION

One of the key aspects of machine design is to provide a sufficiently high level of safety, to long-term operation of the equipment. A common problem arising both from structural defects or sometimes with the phenomenon of fatigue cracking. Cracking phenomenon is an irreversible process that occurs when stresses exceed the border limits of the material.

Modern research methods such as complex experimental analysis and computer simulations, a significant effect on reducing unwanted cases of damage mechanisms during long-term operation. Finite element method is a technique that allows to conduct research numerical analysis mechanism designed of components, before final production. By properly prepared model and to define the numerical process, there is a confirmation of the research through experiments on real objects of research.

[*] Politechnika Lubelska, Nadbystrzycka 36, 20-618 Lublin, +48 603 359 217, p.rozylo@pollub.pl
[**] Politechnika Lubelska, Nadbystrzycka 36, 20-618 Lublin, +48 785 221 354, l.wojcik@pollub.pl

The research sample was made on a 3D printer uPrint SE of ABS material. The object of the research was thin-walled plate which was subjected to a stretching process on testing machine Instron 3369.

The work was carried out numerical analysis of fracture at a critical point the object of study, in order to confront with the result obtained by experimenting. The tool, which was used for the preparation of the numerical process was Abaqus 6.14 environment. The main objective of the research was to compare the obtained path of crack propagation and numerical experiments.

In the cited references [4, 5, 11, 12], the authors presents the introduction of the program Abaqus, indicated the approach used in the design of the basic components of simple geometry in static analysis.

In the papers [1, 2, 6–10], the authors deal with the presentation of a general approach to the problem of propagation of cracks occurring in the material. The authors of these works present experimental results obtained on the basis of the characteristics of the process of fracture of polymeric materials.

In [3] the author takes up the subject describe a general approach to a point of the process of fracture mechanics of materials.

Currently, there is a possibility to use substitutes made of polymeric materials instead of traditional materials consumer.

2. MATERIALS AND METHODS

The research part was a copolymer made of ABS materials. The sample for the study has been to design in Solid Edge ST4 environment and printed on a 3D printer uPrint SE. The dimensions of the element are designed according with the following drawing.

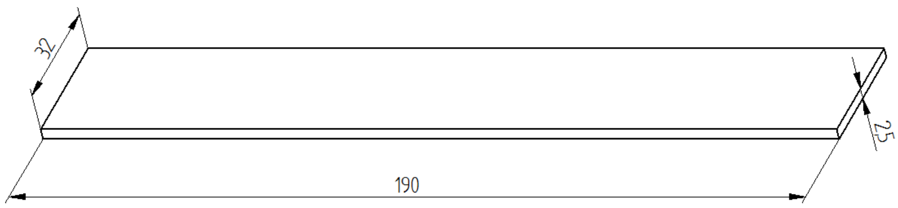


Fig. 1. The dimensions of the element [source: own research]

Model prepared in advance in Solid Edge ST4 software has been imported into the corporate environment uPrint SE 3D printer.

3D printing as opposed to commonly used methods is a method of incremental processing means that the element is produced by the application of successive layers of material. The sample was made on a 3D printer through the use of ABS polymer incremental method. Method of application building material held in by the method of FDM (fused deposition modeling) – modeling liquid thermoplastic.

Initially, this method for the worktable are applied adhesive layer, which is intended to facilitate the finished remove from the printing plate. The next step is to incorporate the constituent material and the support layer. The support layer is used in the created model complex geometry and is inclined with respect to the printing plate at an angle without giving damage effect to the geometry of the performed element. The finished model after the printing process still needs some finishing operations. These include the removal of the selected support material by special detergents and then in a further step is necessary to the working of the outer portion of the object in order to obtain the desired surface quality.

The object of research was characterized by a consistent material properties for the data ABS material. Characteristics of the material necessary to carry out the numerical process based on data spring. Any characteristics of the material shown in the table below.

Tab. 1. Data material of ABS [14]

ABS material	
Young's Modulus [MPa]	1700
Poisson's Ratio	0.38
Yield Strength [MPa]	31
Tensile Strength at Break [MPa]	70

Research station to implement the prepared sample tensile testing machine was Instron 3369. Dual Column Testing Systems is able to perform the processes of compression and tensile test with a maximum force of 50 kN.

The sample was mounted in the jaws of the tensile, symmetrically at a distance of 40 mm from the both ends. The same study was based on a static tensile member, to break the sample. The process led to permanent stretching of separations between the fibers of the material.

Progressive separation of the fibers has consistently led to the destruction of the test piece. The shape of the resulting cracks had stepped characteristics. Area cracking was observed in the area of the mounting location of the sample. Testing machine at work was equipped with a jaw for the stretching process.

Test stand with instrumentation and 3D printer on which the sample was prepared, shown in Figure 2.

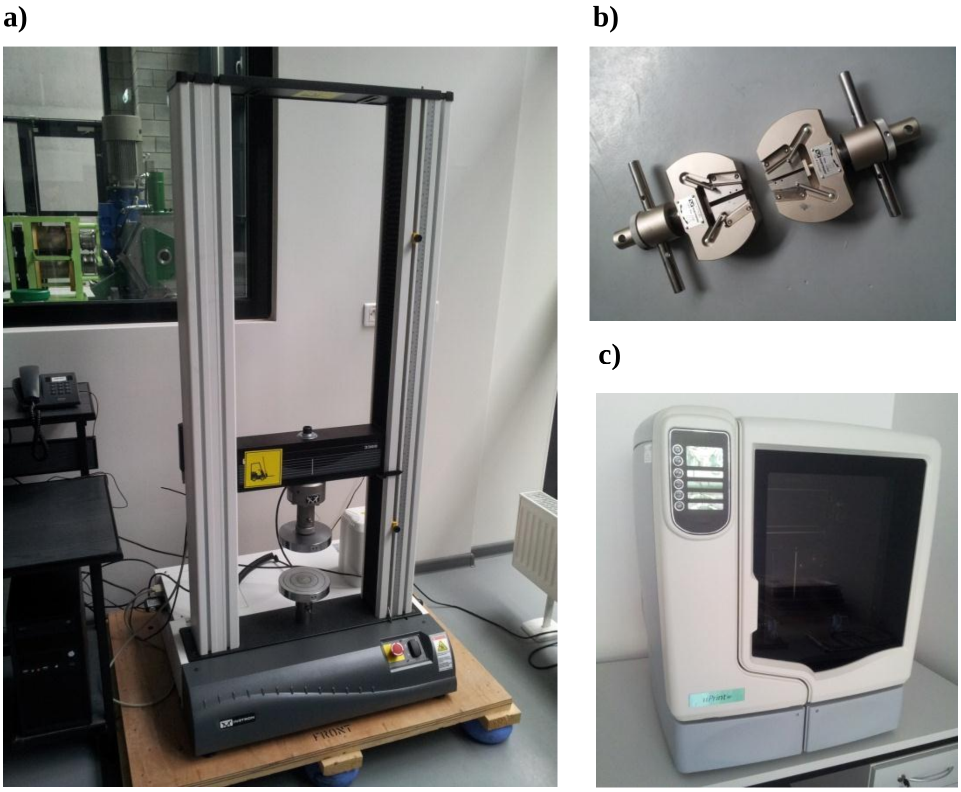
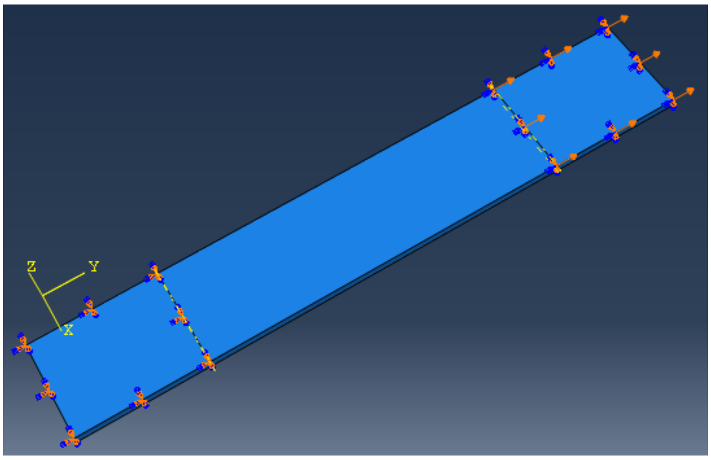


Fig. 2. Test stand: a) Machine Instron 3369, b) Manual jaws Instron 50 kN, c) 3D printer uPrint SE [source: own research]

The next stage of the research was carried out numerical analysis using the finite element method. For this purpose a number of test environment Abaqus 6.14. The program was prepared as part of the rigid body, according to the data in the material element made on a 3D printer. Element had well-defined boundary conditions (fully mapping the actual mounting material in testing machine). One piece of component was fully fixed and the second suffered forced displacement of 1 mm in the direction of Y axis.

The FEM model of type C3D8R tetrahedral elements with a reduced number of integration points. Reduced integration is a technique for removing the impact of the blocking effect of finite elements (reduction of false forms of shape distortion) [13]. Boundary conditions and FEM mesh is shown in Figure 3.

a)



b)

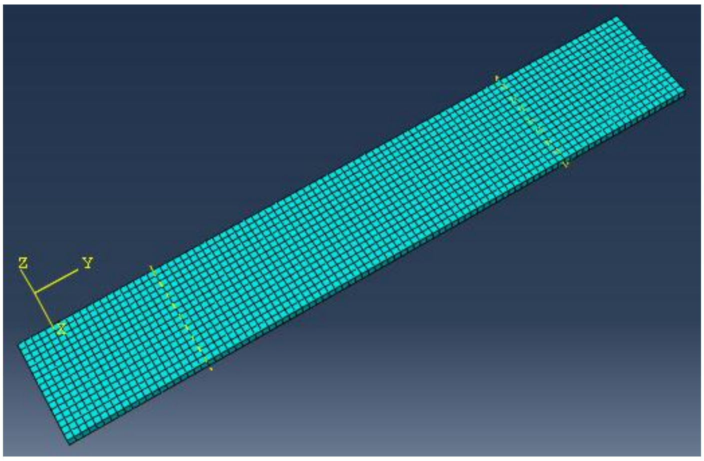


Fig. 3. Numerical model: a) Boundary conditions, b) Mesh [source: own research]

The total number of elements in the mesh was over 1500, with the amount equal to 3264 nodes.

The correctness of the course to simulate real phenomena using numerical methods is dependent on the correct definition of the boundary conditions, the knowledge required for the process material properties and the ability to test the application of the interaction between objects.

The process of crack prepared according to the method xFEM. This method allows for the implementation of crack propagation.

3. RESULTS

FEM analysis allowed us to obtain the stress distribution in the test mechanism. As part of an experiment performed on a testing machine test result was obtained in the form of a broken element.

Comparative destruction of the element by numerical experiments and tests made it possible to observe a similar process of crack propagation in both cases.

Computer simulation of yield showed a more than tenfold, with respect to the base value of R_e = 31 MPa, at which the material permanently burst [14].

Progressive process of permanent separation of the fiber material obtained through simulation studies constantly confronted with the result of progressive cracking on the road made the experiment. The test results of material damage are shown in Figure 4.

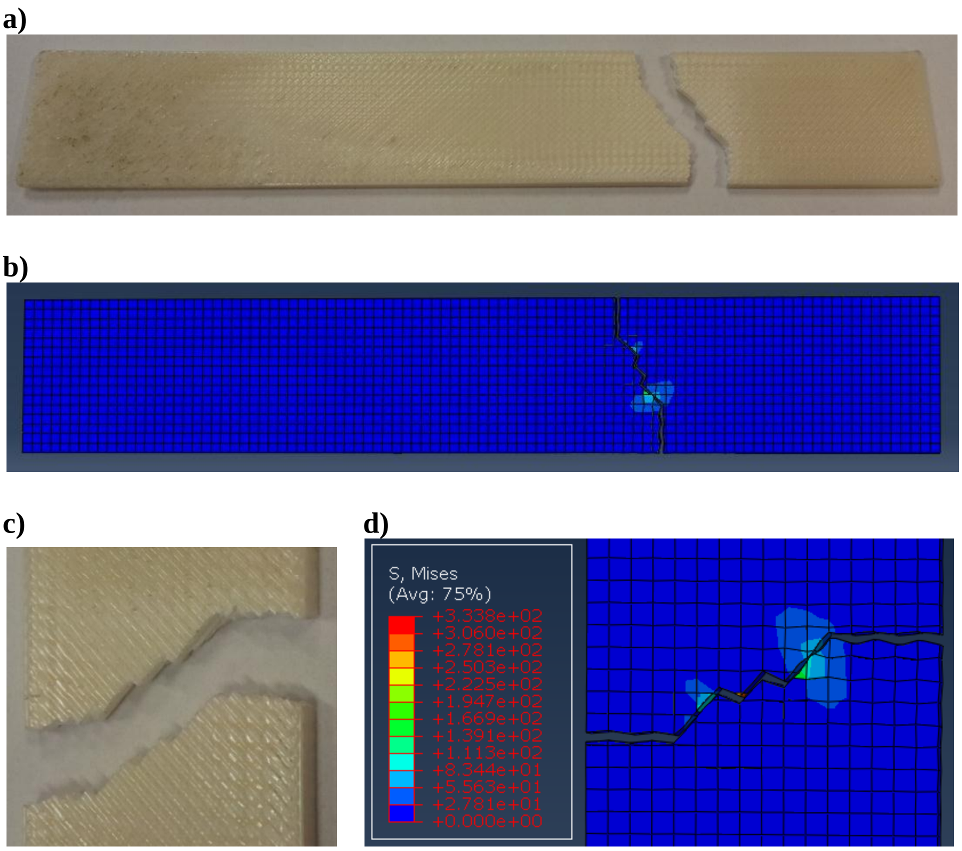


Fig. 4. The comparative results of numerical analysis and experimental study: a) real model after fracture, b) numerical model after analyzing xFEM element, c) the shape of the crack obtained through experience, d) the shape of the crack obtained through FEM [source: own research]

Simulation initiate the process of stretching the sample allowed to obtain form the shape of cracks similar to the one which resulted from previously conducted experiments. The reduced stresses, which have been fixed by separation of all material of the fibers was less than 334 MPa. In almost five times exceeded destructive stress of the material, which amount to 70 MPa there was a total destruction of the sample.

Obtained through computer simulation visualization of material separation, an idealized case of actual rupture results obtained after the stretching process on a laboratory.

4. CONCLUSIONS

With respect to the research it is possible to draw the following conclusions:

- FEM analysis results showed exceeded both the yield strength and the strength of the material, leading to a permanent separation of fibers,
- the shape of cracks obtained by computer analysis is similar to that obtained by the experimental,
- may rupture characteristics obtained stepped mirror symmetrically relative to the axis of the test object,
- analysis of the occurring stress levels appeared sensitive area of the sample in the process of separating material in the form of real-time crack propagation.

Application of FEM to destructive testing in the elements subjected to tensile loading, allows for visualization of the state of effort and permanent separation of fibers prepared numerical models, which nowadays is indispensable in engineering applications.

Observation of crack propagation behavior in relation to the load exerted on the element during operation, allows the proper selection of the material from which it is to be made for future use. Numerical methods allow to predict crack propagation process, thanks to which without prior preparation elements may be beneficial for improving the structural properties.

Observation of the shape of the crack obtained material, in a similar type of analysis gives opportunities to prevent cracking processes by the proper preparation of the sample.

A wide range of research opportunities through experience and performed numerical simulations greatly improves operating conditions studied objects. Computer simulations of physical processes have the potential to future-proof machine optimization methods.

REFERENCES

[1] BOHATKA T. J., MOET A.: *Crack layer analysis of fatigue crack propagation in ABS polymer*. Journal of Materials Science, vol. 30, 1995, pp. 4669–4675.

[2] HERTZBERG R. W., NORDBERG H.: *Fatigue crack propagation in polymeric materials.* Journal of Materials Science, vol. 5, 1970, pp. 521–526.

[3] JONAK J.*: Zagadnienia mechaniki pękania i skrawania materiałów.* Lublin: Politechnika Lubelska, 2010, pp. 90–95.

[4] KĄKOL W., ŁODYGOWSKI T.: *Metoda elementów skończonych w wybranych zagadnieniach mechaniki konstrukcji inżynierskich*. Politechnika Poznańska, 2003.

[5] KLEIBER M.: *Wprowadzenie do metody elementów skończonych*. Biblioteka Mechaniki Stosowanej IPPT PAN, PWN, Warszawa-Poznań, 1985.

[6] MEGEL M., KUMOSA L., ELY T., ARMENTROUT D.: *Initiation of stress corrosion cracking in unidirectional glass/polymer composite materials*. Composites Science & Technology, vol. 61, 2001, pp. 231–246.

[7] MOES N., BELYTSCHKO T.: *Extended finite element method for cohesive crack growth.* Engineering Fracture Mechanics, USA: Northwestern University, 2001.

[8] RAMSTEINER F., ARMBRUST T.: *Fatigue crack growth in polymers*. Polymer Testing, vol. 20, 2001, pp. 321–327.

[9] TRUSS R. W., CHADWICK G. A.: *The tensile deformation behavior of a transparent ABS polymer.* Journal of Materials Science, vol. 11, 1976, pp. 1385–1392.

[10] TURNBULL A., MAXWELL A. S., PILLAI S.: *Comparative assessment of slow strain rate, 4-pt bend and constant load test methods for measuring environment stress cracking of polymers*. Polymer Testing, vol. 19, 2000, pp. 117–129.

[11] VENKATESH C.: *Getting Started with Abaqus* – Workbook 0: User Interface and Modeling Overview, 2011.

[12] VENKATESH C.: *Getting Started With Abaqus* – Workbook 1: Linear Static Analyses and Basic Mesh Generation, 2012.

[13] ZIENKIEWICZ O. C., TAYLOR R. L.: *Finite Element Method (5th Edition)* vol. 2, Solid Mechanics, 2000, Elsevier.

[14] Solid Edge Documentation.